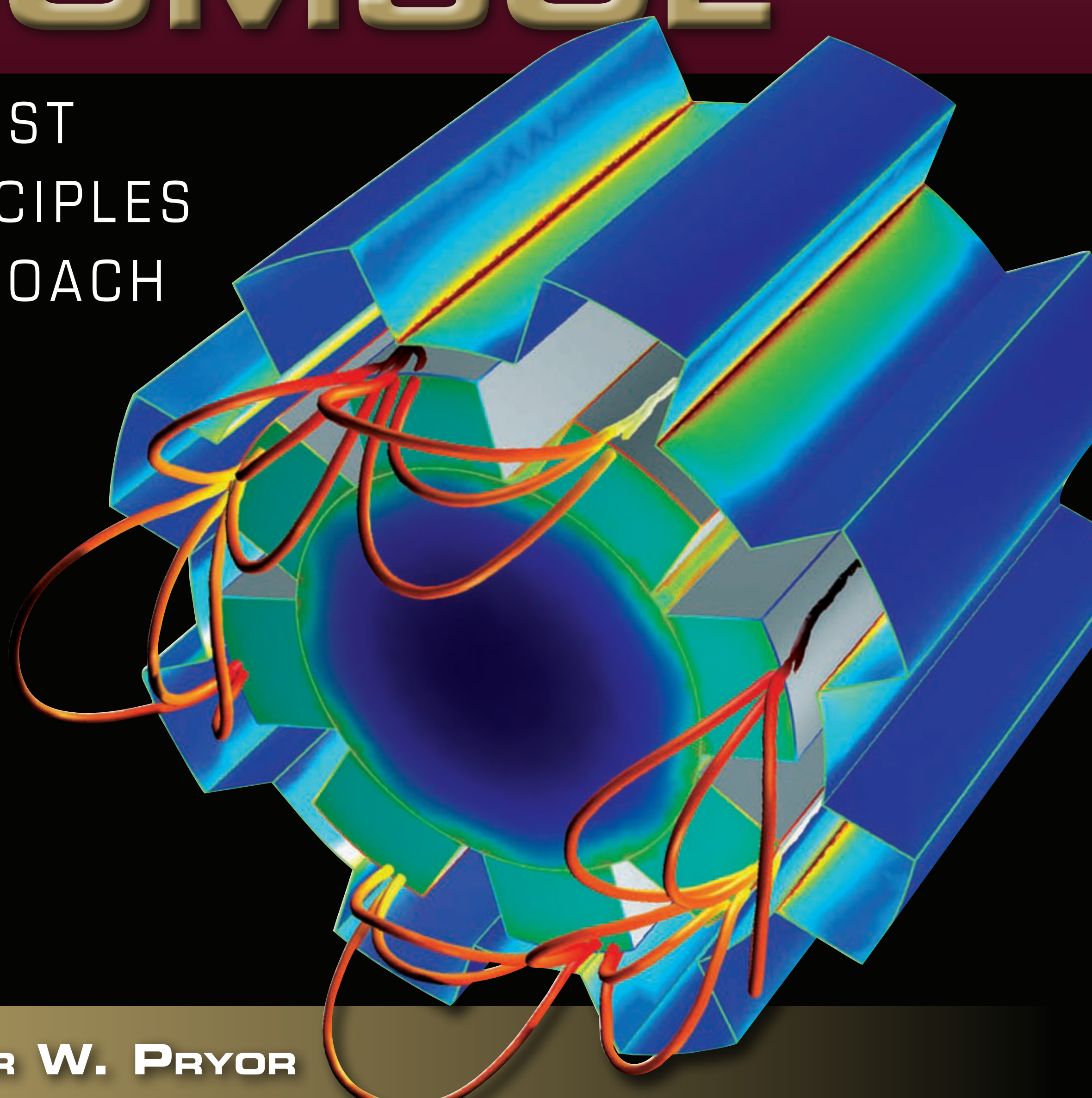
Multiphysics Modeling using COMSOL®

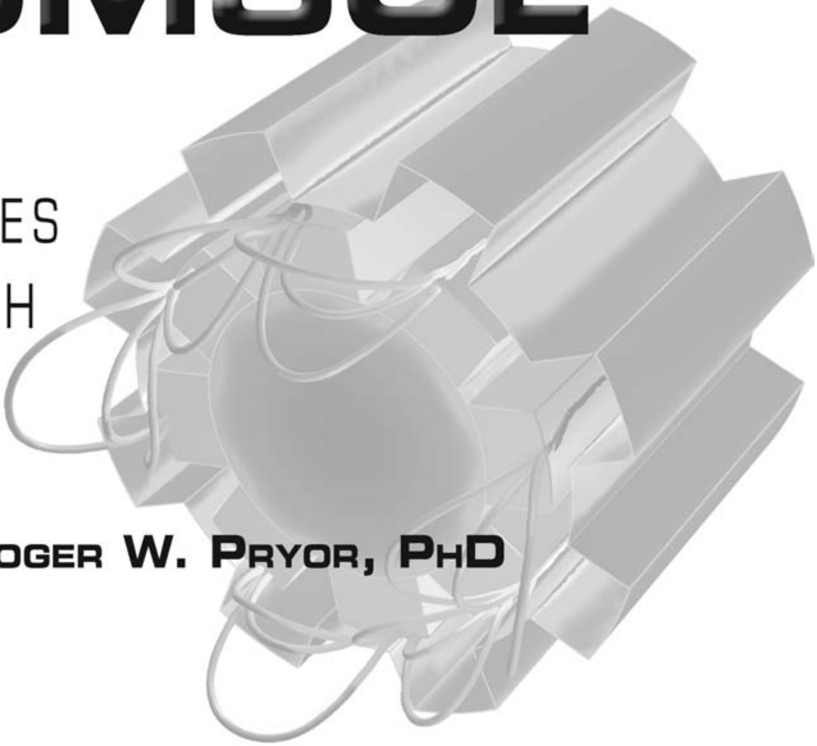
A First Principles Approach



Roger W. Pryor

Multiphysics Modeling using COMSOL®

A First Principles Approach



Roger W. Pryor, PhD

JONES AND BARTLETT PUBLISHERS
Sudbury, Massachusetts
BOSTON TORONTO LONDON SINGAPORE

World Headquarters
Jones and Bartlett Publishers
40 Tall Pine Drive
Sudbury, MA 01776
978-443-5000
info@jbpub.com
www.jbpub.com

Jones and Bartlett Publishers
Canada
6339 Ormindale Way
Mississauga, Ontario
L5V 1J2
Canada

Jones and Bartlett Publishers
International
Barb House, Barb Mews
London W6 7PA
United Kingdom

Jones and Bartlett's books and products are available through most bookstores and online booksellers. To contact Jones and Bartlett Publishers directly, call 800-832-0034, fax 978-443-8000, or visit our website, www.jbpub.com.

Substantial discounts on bulk quantities of Jones and Bartlett's publications are available to corporations, professional associations, and other qualified organizations. For details and specific discount information, contact the special sales department at Jones and Bartlett via the above contact information or send an email to specialsales@jbpub.com.

Production Credits
Publisher: David Pallai
Editorial Assistant: Molly Whitman
Production Director: Amy Rose
Associate Production Editor: Tiffany Sliter
Associate Marketing Manager: Lindsay Ruggiero
V.P., Manufacturing and Inventory Control: Therese Connell
Composition: Glyph International
Cover Design: Scott Moden
Cover Image: © Photo courtesy of COMSOL, Inc. (www.comsol.com)
Printing and Binding: Malloy, Inc.
Cover Printing: Malloy, Inc.

Library of Congress Cataloging-in-Publication Data
Pryor, Roger W.
Multiphysics modeling using COMSOL : a first principles approach / Roger W. Pryor.
p. cm.
Includes index.
ISBN-13: 978-0-7637-7999-3 (casebound)
ISBN-10: 0-7637-7999-7 (casebound)
1. Physics—Data processing. 2. Physics—Mathematical models. 3. Physics—Computer simulation. 4. Engineering—Data processing. 5. Engineering—Mathematical models. 6. Engineering—Computer simulation. 7. COMSOL Multiphysics. I. Title.
QC52.P793 2010
530.15'18—dc22
2009033347

6048

Printed in the United States of America
13 12 11 10 09 10 9 8 7 6 5 4 3 2 1

Contents

Preface

The purpose of this book is to introduce hands-on model building and solving with COMSOL® Multiphysics® software to scientists, engineers, and others interested in exploring the behavior of different physical device structures on a computer, before actually going to the workshop or laboratory and trying to build whatever it is.

The models presented in this text are built within the context of the physical world (applied physics) and are explored in light of first principles analysis techniques. As with any other method of problem solution, the information contained in the solutions from these computer simulations is as good as the materials coefficients and the fundamental assumptions employed in building the models.

The primary advantage in combining computer simulation and first principles analysis is that the modeler can try as many different approaches to the solution of the same problem as needed to get it right (or at least close to right) in the workshop or laboratory the first time that device components are fabricated.

■ Acknowledgments

I would like to thank David Pallai of Jones and Bartlett Publishers for his ongoing encouragement in the completion of this book. I would also like to thank the many staff members of COMSOL, Inc., for their help and encouragement in completing this effort.

I would especially like to thank my wife, Beverly E. Pryor, for the many hours that she spent reading the manuscript and verifying the building instructions for each of the models. Any errors that remain are mine and mine alone.

Roger W. Pryor, Ph.D.

Introduction

COMSOL® Multiphysics® software is a powerful finite element (FEM), partial differential equation (PDE) solution engine. The basic COMSOL Multiphysics software has eight add-on modules that expand the capabilities of the basic software into the following application areas: AC/DC, Acoustics, Chemical Engineering, Earth Science, Heat Transfer, MEMS, RF, and Structural Mechanics. The COMSOL Multiphysics software also has other supporting software, such as the CAD Import Module and the Material Library.

In this book, scientists, engineers, and others interested in exploring the behavior of different physical device structures through computer modeling are introduced to the techniques of hands-on building and solving models through the direct application of the COMSOL Multiphysics software, the AC/DC Module, the Heat Transfer Module, and the RF Module. Chapter 9 explores the use of perfectly matched layers (PML) in the RF Module. The final technical chapter (Chapter 10) explores the use of the bioheat equation in the Heat Transfer Module.

The models presented here are built within the context of the physical world (applied physics) and are presented in light of first principles analysis techniques. As with any other methodology of problem solution, the information derived from the modeling solutions through use of these computer simulations is only as good as the materials coefficients and the fundamental assumptions employed in building the models.

The primary advantage derived from combining computer simulation and first principles analysis is that the modeler can try as many different approaches to the solution of the same problem as needed to get it right (or at least close to right) in the workshop or laboratory before the first device components are fabricated and tested. The modeler can also use the physical device test results to modify the model parameters and arrive at a final solution more rapidly than by simply using the cut-and-try methodology.

■ Chapter Topics

This book comprises ten technical chapters. Its primary focus is to demonstrate to the reader the hands-on technique of model building and solving. The COMSOL Concepts and Techniques are shown in Figure 1. The COMSOL modules employed in the various

	Chapter									
Concept/Technique	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**
1D PDE modeling	•		•							
2D Axisymmetric coordinates						•				
2D Axisymmetric modeling					•					
2D Modeling				•					•	•
3D Modeling								•		
Animation			•	•	•	•	•		•	•
Azimuthal inductive heating						•				
Bioheat equation										•
Boundary conditions			•							
Boundary integration					•					
CAD drawings (geometry objects), export and import					•	•				
Conductive media Pc				•	•	•	•	•		
Constants, import and export					•					
Cross-section plot				•		•		•		
Cylindrical coordinates					•					
Deformed mesh – moving mesh (ALE)				•			•			
Domain plot parameter							•			
Electromagnetics				•						
Electrostatic potentials								•		
Electro-thermal coupling						•				
Floating contacts				•						
Free mesh parameters				•	•	•				
Geometric assembly (pair creation across a boundary)							•			
Global equations							•			
Heat transfer coefficients	•				•					
Imbalance-offset geometry				•						
Induction heating						•				
In-plane electric currents							•			
In-plane te waves									•	
Iterative solver								•		
Lagrange parameters										•
Laplacian operator								•		
Magnetostatic modeling								•		
Materials library	•	•			•		•			
Maximum element size				•	•	•				
Mesh mapping							•			
Mixed-materials modeling						•				
Mixed-mode modeling						•				
Opaque and transparent thermally conductive materials					•					
Ordinary differential equation (ODE)							•			
Parametric solutions			•		•					
Perfectly matched layers									•	
Periodic point conditions							•			
Perpendicular induction currents							•			
Polyline drawings							•			
Quadrilateral mesh (Quad)				•	•	•				
Quasi-static solutions					•					
Reference frame							•			
Rotating machinery							•			
Scalar expressions					•	•				
Scalar variables						•				
Static solutions					•					
Subdomain mesh				•	•	•				
Suppress subdomain									•	
Surface integrals					•					
Thin layer approximation								•		
Thin layer subdomain								•		
Time-harmonic analysis							•			
Transient analysis				•		•				
Triangular mesh				•	•	•				
Weak constraints				•						

FIGURE 1 COMSOL Concepts and Techniques

Module	1	2	3	4	5	6	7	8	9	10
Basic	•	•	•	•	•	•	•	•	•	•
AC/DC				•		•	•	•		
Heat Transfer					•					•
Materials Library		•								

FIGURE 2 COMSOL Modules Employed

models in specific chapters are shown in Figure 2, and the physics concepts and techniques employed in the various models in specific chapters are shown in Figure 3.

These grids link the overall presentation of this book to the underlying modeling, mathematical, and physical concepts. In this book, in contrast to some other books with which the reader may be familiar, key ancillary information, in most cases, is contained in the notes.

NOTE Please be sure to read, carefully consider, and apply, as needed, each note.

Chapter 1. Modeling Methodology

Chapter 1 begins the introduction to the modeling process by discussing the fundamental considerations involved: the hardware (computer platform), the coordinate systems (physics), the implicit assumptions (lower dimensionality considerations), and first principles analysis (physics). Three relatively simple 1D models are presented, built, and solved for comparison: one-pane, two-pane, and three-pane thermal insulation window structures. Comments are also included on common sources of modeling errors.

Chapter 2. Materials and Databases

Chapter 2 briefly introduces three sources of materials properties data: the COMSOL Material Library, MatWeb, and the PKS-MPD.

The COMSOL Material Library is a module that can be added to the basic COMSOL Multiphysics software package to expand the basic library that is already included. It contains data on approximately 2500 materials, including elements, minerals, soil, metal alloys, oxides, steels, thermal insulators, semiconductors, and optical materials. Each material can have up to 27 defined properties. Each of those defined properties is available as a function of temperature.

MatWeb is an online searchable subscription materials properties data source. MatWeb has three classes of access: Unregistered (free limited feature access), Registered Member (free expanded feature access), and Premium Member (fee-based

	Chapter									
Physics Concepts	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**
AC induction						•				
Anisotropic conductivity				•						
Antennas										•
Bioheat equation										•
Boltzmann thermodynamics					•					
Complex AC theory							•			
Complex impedance							•			
Concave mirror									•	
Coulomb gauge								•		
Dielectric lenses									•	
Distributed resistance								•		
Electrical impedance theory							•			
Electrochemical polishing				•						
Electrostatic potentials in different geometric configurations								•		
Faraday's law				•						
First Estimate Review	•							•		
Foucault (eddy) currents						•				
Fourier's law						•				
Free-space permittivity								•		
Good first approximation	•					•				
Hall effect				•						
Hard and soft nonlinear magnetic materials							•			
Heat conduction theory	•				•					
Helmholtz coil								•		
Information transmission			•							
Insulated containers					•					
Joule heating					•	•				
Lorentz force				•						
Magnetic field				•						
Magnetic permeability								•		
Magnetic vector potential								•		
Magnetostatics								•		
Maxwell's equations					•			•	•	
Mechanical to electrical energy conversion							•			
Microwave irradiation										•
Newton's law of coding					•					
Ohm's law						•	•	•		
Optical (laser) irradiation										•
Pennes's equation										•
Perfectly matched layers: 2D planar, 3D cartesian cylindirical and Spherical									•	
Perfusion										•
Planck's constant					•					
Power transmission grids, AC and DC							•			
Power transmission, AC and DC							•			
Reactance							•			
Semiconductor dual carrier types				•						
Skin depth							•			
Soliton waves			•							
Telegraphs equation			•							
Thin layer resistance								•		
Vacuum					•					
Vector dot product current (K*Nj_DC)				•						

FIGURE 3 Physics Concepts and Techniques

access to all features, plus selected data storage and modeling software formatted data export). MatWeb has 69,000 data sheets for materials, including plastics, metals, ceramics, semiconductors, fibers, and various other commercially available materials.

PKS-MPD (Pryor Knowledge Systems—Materials Properties Database) is a new searchable materials properties database with data on more than 4000 materials, including elements, minerals, soil, metal alloys, oxides, steels, thermal insulators, semiconductors, optical materials, and biomaterials (tissue). Each material can have up to 43 defined properties. Each of those defined properties is associated with the temperature of measurement and the frequency of measurement. The collection of defined properties for each materials property datum is exportable in a format suitable for use with the COMSOL Multiphysics software.

■ Chapter 3. 1D Modeling

The first half of Chapter 3 models the 1D KdV equation and two variations. The KdV equation is a powerful tool that is used to model soliton wave propagation in diverse media (e.g., physical waves in liquids, electromagnetic waves in transparent media). It is easily and simply modeled with a 1D PDE mode model.

The second half of Chapter 3 models the 1D telegraph equation and two variations. The telegraph equation is a powerful tool that is used to model wave propagation in diverse transmission lines. It can be used to thoroughly characterize the propagation conditions of coaxial lines, twin pair lines, and microstrip lines, among other things. The telegraph equation is easily and simply modeled with a 1D PDE mode model.

■ Chapter 4. 2D Modeling

The first half of Chapter 4 models the 2D electrochemical polishing model. This model is a powerful tool that can be used to model surface smoothing for diverse projects (e.g., microscope samples, precision metal parts, medical equipment and tools, large and small metal drums, thin analytical samples, vacuum chambers).

The second half of Chapter 4 models the 2D Hall effect. The 2D Hall Effect model is a powerful tool that can be used to model Hall effect magnetic sensors for sensing fluid flow, rotating and linear motion, proximity, current, pressure, and orientation.

■ Chapter 5. 2D Axisymmetric Modeling

The first half of Chapter 5 models three 2D axisymmetric cylinder conduction models. From a comparison of the three models, it can be readily observed that the presence of a vacuum cavity significantly reduces the rate of heat flow through the model and raises the equilibrium temperature at the surface receiving the heat flux.

The second half of Chapter 5 models three 2D axisymmetric thermos container models. From a comparison of the three models, it can be readily observed that the presence of a vacuum cavity significantly reduces the rate of heat flow through the model and the associated heat loss.

■ Chapter 6. 2D Simple Mixed-Mode Modeling

The first half of Chapter 6 models three 2D resistive heating models. These models are more illustrative of the mixed-mode modeling concept than they are directly amenable to the comparison of calculated values. They present different examples of the diversity of applied scientific and engineering model designs that can be explored using electro-thermal coupling and transient analysis. These models also demonstrate the significant power of relatively simple physical principles, such as Ohm's law and Joule's law.

The second half of Chapter 6 models three 2D axisymmetric inductive heating models. These models demonstrate the difference in level of complexity between single-coil and multi-coil models. In the Inductive_Heating_1 model, the concept of inductively produced heating is introduced. In the Inductive_Heating_2 model, the concept of inductively produced heating is applied to a practical application (a heated crucible) so as to present one example of the diverse applied scientific and engineering model designs that can be explored using electro-thermal coupling and transient analysis. In the Inductive_Heating_3 model, the crucible is filled with a commonly used metal for melting.

These models are examples of the good first approximation type of model. In other words, they demonstrate the significant power of relatively simple physical principles, such as Ohm's law and Joule's law, when applied in the COMSOL Multiphysics modeling environment. They could, of course, be modified by the addition of calculations, insulating materials, and heat loss through convection, among other changes.

■ Chapter 7. 2D Complex Mixed-Mode Modeling

The first third of Chapter 7 introduces two 2D electric impedance sensor models: basic and advanced. Those models employ high-frequency currents—1 MHz alternating currents AC—to explore the differential impedance within a body of material in a noninvasive fashion. Such currents are applied to the material of the modeled body to locate volumes that differ in impedance from the impedance of the bulk material by monitoring the local impedance.

The basic version models the location of a fixed-volume impedance difference. The advanced version models the location of a fluctuating difference volume, as might be seen in a medical application measuring lung function. 2D electric impedance tomography research is currently exploring the application of this type of impedance

sensing measurement technology to the detection of breast cancer, lung function, brain function, and numerous other areas.

The second third of Chapter 7 introduces two 2D AC generator models: static and transient. These models generate low-frequency (60 Hz) current and voltage, as would be typically found on the power transmission grid. They demonstrate the use of both hard (not easily magnetized) and soft (easily magnetized) nonlinear magnetic materials in the construction of rotating machines for the conversion of mechanical energy to electrical energy.

The last third of Chapter 7 introduces two 2D AC generator sector models: static and transient. These models generate low-frequency (60 Hz) current and voltage, as would be typically found on the power transmission grid. They demonstrate the use of both hard (not easily magnetized) and soft (easily magnetized) nonlinear magnetic materials in the construction of a rotating machine for the conversion of mechanical energy to electrical energy. An ordinary differential equation (ODE) is incorporated into the sector model to handle the torque-related aspects of the model calculations.

Chapter 8. 3D Modeling

The first third of Chapter 8 models the 3D thin layer resistance model, thin layer approximation, and the thin layer resistance model, thin layer subdomain. The first model employs the thin layer approximation to solve a model by replacing the center domain with a contact-resistance identity pair. Such an approximation has broad applicability. It is important to note that the use of the thin layer approximation is applicable to any problem in which flow is described by the divergence of a gradient flux (e.g., diffusion, heat conduction, flow through porous media under Darcy's law).

The application of the thin layer approximation is especially valuable to the modeler when the differences in domain thickness are so great that the mesh generator fails to properly mesh the model or creates more elements than the modeling platform can handle ("run out of memory" problem). In those cases, this approximation may enable a model to be solved that would otherwise fail.

A direct comparison is made of the model solutions by comparing the results obtained from the cross-section plots. As seen from the examination of those plots, the only substantial difference between the two solutions is the electrical potential difference across subdomain 2 (the thin layer). Thus the modeler can choose the implementation that best suits his or her system and time constraint needs, without suffering excessive inaccuracies based on the approximation method.

The second third of Chapter 8 introduces the 3D electrostatic potential model. This modeling technique demonstrates one of the methods that can be used by the modeler to explore electrostatic potentials in different geometric configurations. It can be applied to both scientific and engineering applications (e.g., ranging from X-ray

tubes and particle accelerators to paint sprayers and dust precipitators). The 3D_ESP_2 model is typical of those that might be found in a particle beam analyzer or a similar engineering or scientific device.

The last third of Chapter 8 models the 3D magnetic field of a Helmholtz coil. This model demonstrates the magnetic field uniformity of a Helmholtz coil pair. This magnetostatic modeling technique can be applied to a diverse collection of scientific and engineering applications (e.g., ranging from magnetometers and Hall effect sensors to biomagnetic and medical studies).

A related model, the 3D magnetic field of a Helmholtz coil with a magnetic test object, demonstrates the magnetic field concentration when a high relative permeability object lies within the field of the Helmholtz coil. This magnetostatic modeling technique can be applied to a diverse collection of scientific and engineering test, measurement, and design applications.

■ Chapter 9. Perfectly Matched Layer Models

The first half of Chapter 9 introduces the 2D dielectric lens models, with and without perfectly matched layers (PMLs). [The PML model best approximates a free space environment (no reflections).] Comparison is made between the two models. The differences in the electric field, z-component visualizations between the PML and no-PML models amount to approximately 2%. Depending on the nature of the problem, such differences may or may not be significant. What these differences show the modeler is that he or she needs to understand the application environment well so as to build the best model. For other than free space environments, the modeler needs to determine the best boundary condition approximation using standard practices and a first principles approach. Always do a first principles analysis of the environment before building the model.

The second half of Chapter 9 introduces the 2D concave mirror models, with and without PMLs. There are only small differences in the electric field, z-component visualizations between the PML and no-PML models for the concave mirror. This lack of large differences between the PML and no-PML models again shows the modeler that he or she needs to understand the relative importance of the modeled values to evaluate the application and the application environment so as to build the best model.

■ Chapter 10. Bioheat Models

The bioheat equation plays an important role in the development and analysis of new therapeutic medical techniques (e.g., killing of tumors). If the postulated method raises the local temperature of the tumor cells without excessively raising the temperature of the normal cells, then the proposed method will probably be successful.

The results (estimated time values) from the model calculations will significantly reduce the effort needed to determine an accurate experimental value. The guiding principle needs to be that tumor cells die at elevated temperatures. The literature cites temperatures that range from 42 °C (315.15 K) to 60 °C (333.15 K).

The first half of Chapter 10 models the bioheat equation as applied with a photonic heat source (laser). The second half of the chapter models the bioheat equation as applied with a microwave heat source.

NOTE Executable copies of each model and related animations are available in **full color** on the accompanying DVD.

---

1 Modeling Methodology

In This Chapter

■ Guidelines for New COMSOL® Multiphysics® Modelers

Hardware Considerations

There are two basic rules to selecting hardware that will support successful modeling. First, new modelers should be sure to determine the minimum system requirements that their version of COMSOL® Multiphysics® software needs before borrowing or buying a computer to run their new modeling software. Second, these new modelers should run their copy of COMSOL Multiphysics software on the best platform with the highest processor speed and the most memory obtainable: The bigger and faster, the better. It is the general rule that the speed of model processing increases directly as a function of the processor speed, the number of platform cores, and the available memory.

NOTE The number of platform cores is equal to the number of coprocessors designed into the computer (e.g., one, two, four, eight, . . .).

The platform that this author uses is an Apple® Mac Pro®, running Mac OS X® version 10.5.x, and also running Parallels Desktop® 3.x with Microsoft® Windows XP®.

This Mac Pro has four 3 GHz cores and 16 GB of RAM. This platform, as configured, is more powerful, more versatile, more stable, and more cost-effective than other potential choices. It can handle complex 3D models in short computational times (more speed, more memory)—that is, in minutes instead of the hours that may be required by less powerful systems. This configuration can run any of the 32-bit COMSOL Multiphysics software, when using COMSOL Multiphysics Version 3.4. The Apple hardware is configured for 64-bit processing and will run at the 64-bit rate when using COMSOL Multiphysics Version 3.5. If new modelers desire a different 64-bit operating system than Macintosh OS X, then they will need to choose either a Sun® or a Linux® platform, using UNIX© or a PC with a 64-bit Microsoft Windows operating system.

NOTE The "3 GHz" specification is the operating speed of each of the cores and the "16 GB" is the total shared random access memory (RAM). The "64-bit" refers to the width of a processor instruction.

Once the best available processor is obtained, within the constraints of your budget, install your copy of COMSOL Multiphysics software, following the installer instructions. Once installed, COMSOL Multiphysics software presents the modeler with a graphical user interface (GUI). For computer users not familiar with the GUI concept, information in such an interface is presented primarily in the form of pictures with supplemental text, not exclusively text.

Coordinate Systems

Figure 1.1 shows the default (x-y-z) coordinate orientation for COMSOL modeling calculations. This coordinate system is based on the right-hand rule.

NOTE The right-hand rule is summarized by its name. Look at your right hand, point the thumb up; point your first finger away from your body, at a right angle (90 degrees) to your thumb; and point your second finger at a right angle to the thumb and first finger, parallel to your body. Your thumb represents the z-axis, your first finger represents the x-axis, and your second finger represents the y-axis.

In this right-handed coordinate system, x rotates into y and generates z. If you have a need to convert your model from the x-y-z frame to a Spherical Coordinate frame, then the transformation can be implemented using built-in COMSOL mathematical functions. The x-y-z to spherical coordinate conversion is achieved through the following equations:

Spherical radius (r): $$r = \mathrm{sqrt}(x^2 + y^2 + z^2) \tag{1.1}$$

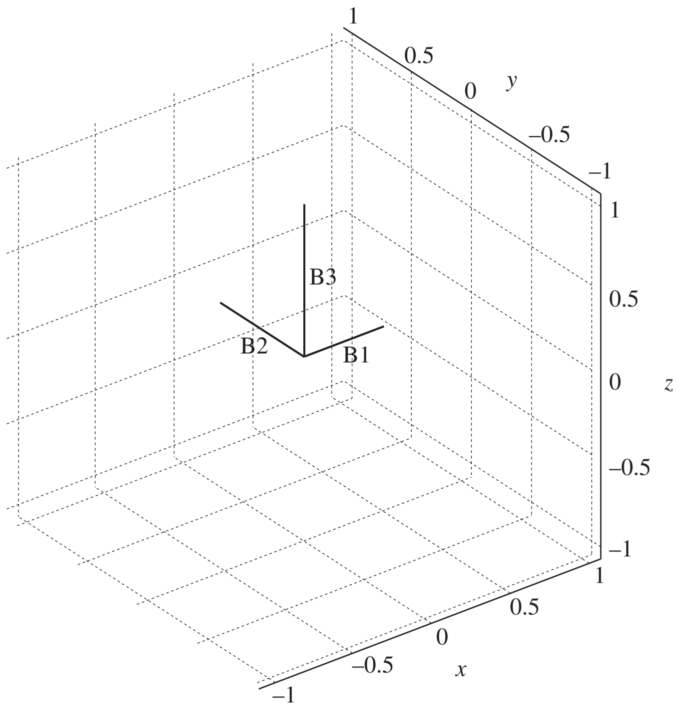


FIGURE 1.1 3D GUI example of the Cartesian coordinate system (*x-y-z*)

$$x\text{-}y \text{ plane rotational angle } \phi \text{ (phi):} \quad \text{phi} = a\tan2(y, x) \tag{1.2}$$

$$x\text{-}z \text{ plane rotational angle } \theta \text{ (theta):} \quad \text{theta} = a\cos(z/r) \tag{1.3}$$

NOTE The built-in function sqrt(argument) indicates that COMSOL Multiphysics will take the positive square root of the argument contained between the parentheses. The built-in function *a* tan2(argument) indicates that COMSOL Multiphysics will convert the argument contained between the parentheses to an angle in radians (in this case, phi). The built-in function *a* cos(argument) indicates that COMSOL Multiphysics will convert the argument contained between the parentheses to an angle in radians (in this case, theta).

To employ these spherical conversion equations in your model, you will need to start COMSOL Multiphysics, select "3D" in the Model Navigator Screen, select the desired Application Mode, and click the OK button. Using the pull-down menu, select Options > Expressions > Scalar Expressions and then enter equations 1.1, 1.2, and 1.3 in the Scalar Expressions window, as shown in Figure 1.2.

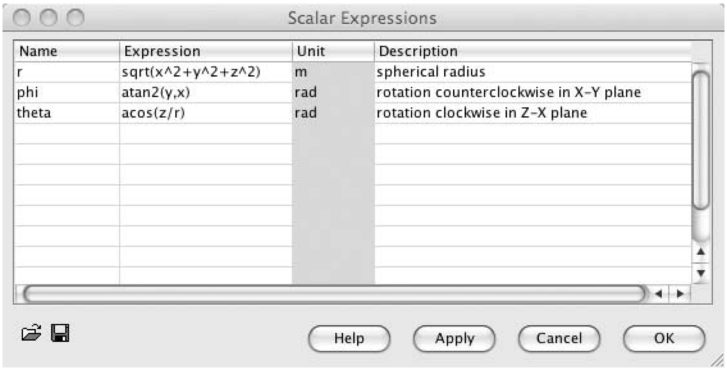


FIGURE 1.2 COMSOL Multiphysics 3D Scalar Expressions window with the spherical coordinate transform equations entered

---

NOTE When a list of operations is presented sequentially (A > B > C > D), the modeler is expected to execute those operations in that sequence in COMSOL.

Once these equations are available in the model, the *x*-*y*-*z* coordinates can be converted as shown in Figures 1.3 and 1.4.

NOTE The rotational sense of an angular transform is determined by viewing the angular rotation as one would view a typical analog clock face. The vector **r** rotating from *x* into *y* for a positive angle phi is counterclockwise, because phi equals zero at the positive *x*-axis. The vector **r** rotating from *z* into *x* for a positive angle theta is clockwise, because theta equals zero at the positive *z*-axis.

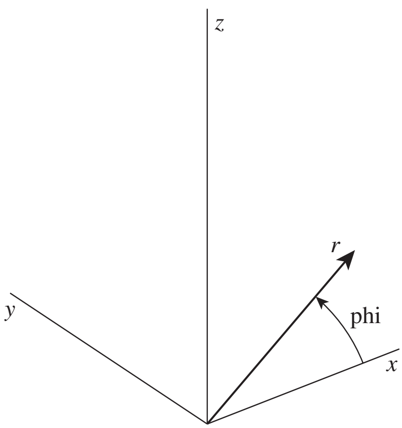


FIGURE 1.3 Spherical coordinate transform angle and rotational sense for (ϕ) phi

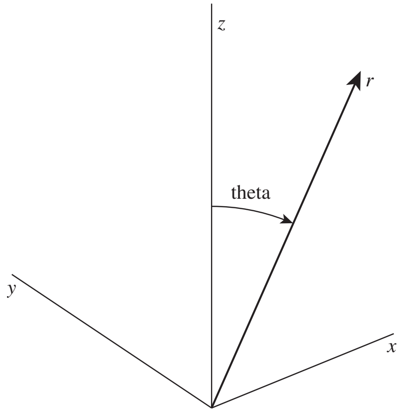


FIGURE 1.4 Spherical coordinate transform angle and rotational sense for (θ) theta

As all potential COMSOL modelers know, the Cartesian (*x*-*y*-*z*) and Spherical Coordinate (*r*-phi-theta) reference frames are not the only coordinate systems that can be used as the basis frame for Multiphysics models. In fact, when you first open the Model Navigator, you are given the option and are required to choose one of the following modeling coordinate systems: 1D, 2D, 3D, Axial Symmetry (1D), or Axial Symmetry (2D). The coordinate system that you choose determines the geometry and specific subgroup of COMSOL Application Modes that can be applied in that selected geometry (geometries).

NOTE An Application Mode is the initial collection of equations, independent variable(s), dependent variable(s), default settings, boundary conditions, and other properties that are appropriate for the solution of problems in that branch of physics (e.g., acoustics, electromagnetics, heat transfer). As indicated by the name "Multiphysics," multiple branches of physics can be applied within each model. Diverse models that demonstrate the application of the multiphysics concept will be explored in detail as this book progresses.

Implicit Assumptions

A modeler can generate a first-cut problem solution as a reasonable estimate, by choosing initially to use a lower-dimensionality coordinate space than 3D (e.g., 1D, 2D Axisymmetric). By making a low-dimensionality geometric choice, a modeler can significantly reduce the total time needed to achieve a detailed final solution for the chosen prototype model. Both new modelers and experienced modelers alike must be especially careful to fully understand the underlying (implicit) assumptions, unspecified conditions, and default values that are incorporated into the model as a result of simply selecting the lower-dimensionality geometry.

NOTE A first-cut solution is the equivalent of a back-of-the-envelope or on-a-napkin solution. Solutions of this type are relatively easy to formulate, are quickly built, and provide a first estimate of whether the final solution of the full problem is deemed to be (should be) within reasonable bounds. Creating a first-cut solution will often allow the modeler to decide whether it is worth the time and money required to create a fully implemented higher-dimensionality (3D) model.

Space, as all modelers know, comes with four basic dimensions: three space dimensions (x-y-z) and one time dimension (t). For example, the four dimensions might be x-y-z-t, or r-ϕ-θ-t. Relativistic effects can typically be neglected, except in cases of high velocity or ultra-high accuracy. Neither of these types of problems will be covered in this book.

NOTE Relativistic effects typically become a concern only for bodies in motion with a velocity approaching that of the speed of light (~3.0 × 108 m/s) or for ultra-high resolution time calculations at somewhat lower velocities.

The types of calculations presented within this book are typically for steady-state models or for relatively low-velocity transient model solutions. Any transient solution model can be solved using a quasi-static methodology.

NOTE In a steady-state model, the controlling parameters are defined as numerical constants and the model is allowed to converge at the equilibrium state defined by the specified constants. In the quasi-static methodology, a model solution to a problem is found by initially treating the model as a steady-state problem. Incrementally modifying the modeling constants then moves the model problem solution toward the desired transient solution.

1D Window Panes Heat Flow Models

Consider, for example, a brief comparison between a relatively simple 1D heat flow model and the identical problem presented as a 3D model. The models considered here are those of a single-pane, dual-pane, or triple-pane window mounted in the wall of a building on a typical winter's day. The questions to be answered are Why use a dual-pane window? and Why use a triple-pane window?

1D Single-Pane Heat Flow Model

Run the COMSOL Multiphysics application. Select "New" and then select "1D" in the Model Navigator. Then select "COMSOL Multiphysics." Select "Heat Transfer" followed by "Conduction" and then "Steady-state analysis." Click OK.

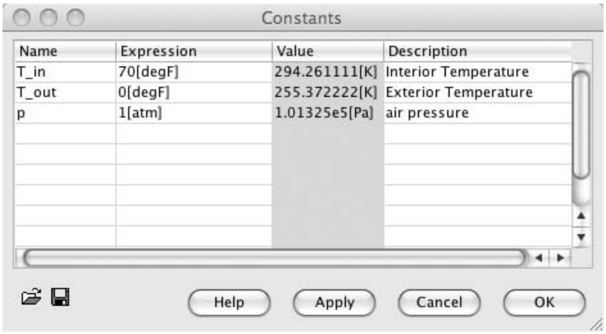


FIGURE 1.5 1D Constants specification window

After the 1D workspace appears, enter the constant values needed for this model: Use the menu bar to select Options > Constants. Enter the following items: T_in tab 70[degF] tab Interior Temperature tab T_out tab 0[degF] tab Exterior Temperature tab p tab 1[atm] (in Version 3.4) or 1.01325e5[Pa] (for earlier versions) tab air pressure tab; see Figure 1.5. Click the Apply button. These entries in the Constants window define the Interior Temperature, the Exterior Temperature, and the air pressure for use in this model. Click on the disk icon in the lower-left corner of the Constants window (Export Variables to File) to save these constants as ModelOC_1D_WP1.txt for use in the comparison models to follow later in this chapter. Click the OK button.

Next, draw a line to represent the thickness of a 1D window pane: Use the menu bar and select Draw > Specify Objects > Line. Enter the following: 0.000 space 0.005; see Figure 1.6. Leave the default Polyline, and click the OK button. Click the Zoom Extents icon in the toolbar.

Once the Zoom Extents icon is clicked, the specified 0.005 m line will appear in the workspace, as shown in Figure 1.7.

After the 0.005 m line has been created in the workspace, use the menu bar and select Physics > Subdomain Settings > 1. The window shown in Figure 1.8 contains the known properties of copper (Cu) as the default materials properties values.

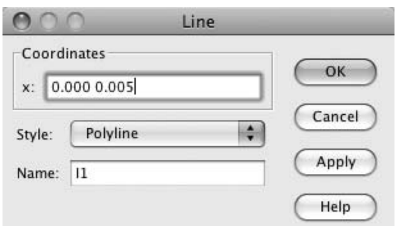


FIGURE 1.6 1D Line specification window

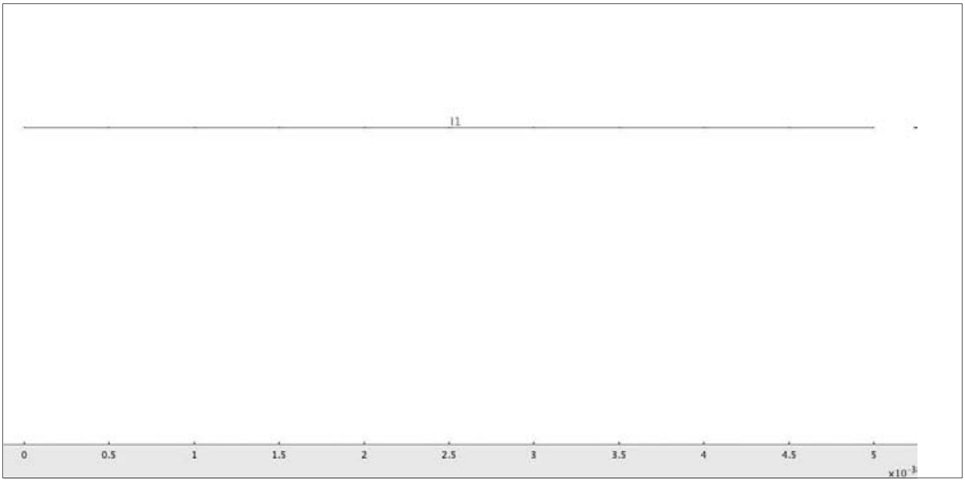


FIGURE 1.7 The 0.005 m line shown in the 1D workspace

NOTE COMSOL Multiphysics software is based on the Finite Element Method (FEM). To ensure that it is as easy as possible to use out of the box for modelers, both new and experienced, COMSOL inserts default materials properties and numerical parameters settings values to avoid singularities and other errors in the calculation of solutions. The modeler will need to verify that all materials and parameter values that are incorporated into their particular models are the appropriate values for the desired solution.

FIGURE 1.8 1D Subdomain Settings window

If copper is not the material of choice, as in this heat transfer model, then the materials property values need to be changed.

NOTE The implicit assumption here would be that the default values in the Subdomain Settings window are the correct values that the modeler needs to build the desired model. Specifically, that assumption would be true only if the modeler were building the heat transfer model using copper. In general, that implicit assumption is not correct. The modeler needs to know before building any new models what the approximate expected values are for the particular properties of the materials selected for use in the model. A number of sources have detailed materials properties values available: Some sources are available at no cost, while other sources have different levels of availability for different fees. Materials properties sources are discussed in Chapter 2.

For this model, click the Load button, and then select "Basic Materials Properties" and "Silica Glass." Click OK. All the appropriate values displayed in the Value/Expression subwindows in the Subdomain Settings window are altered as the new values are loaded from the library. (Silica glass has a thermal conductivity value roughly 0.35% of that of copper.)

Once the thermal conductivity is loaded from the materials properties library, enter T_in in the T_{ext} and $T_{ambtrans}$ windows, as shown in Figure 1.9.

Click the Init button and enter T_in as shown in Figure 1.10. Click OK. Setting the T_in value as the initial temperature of the window pane (subdomain 1) allows for quicker convergence of the model and avoids any singularities.

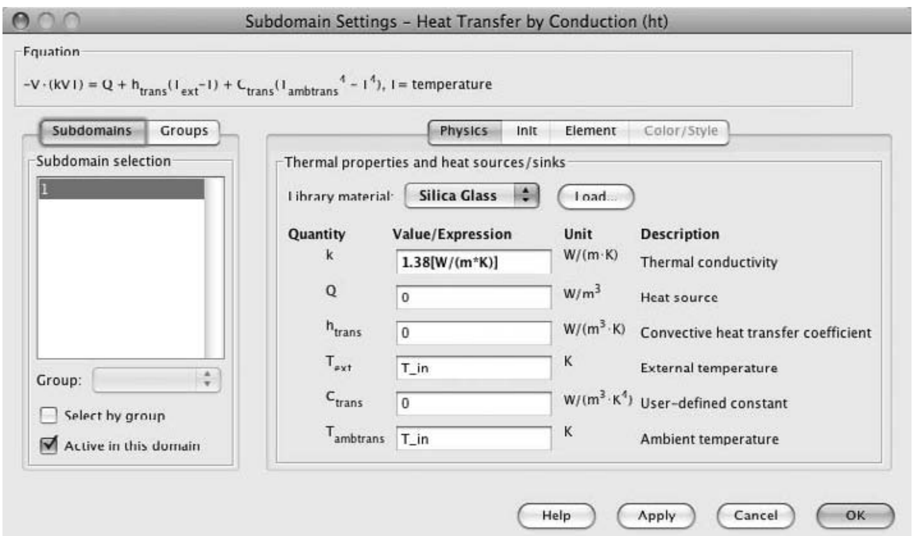


FIGURE 1.9 1D Subdomain Settings Physics window settings

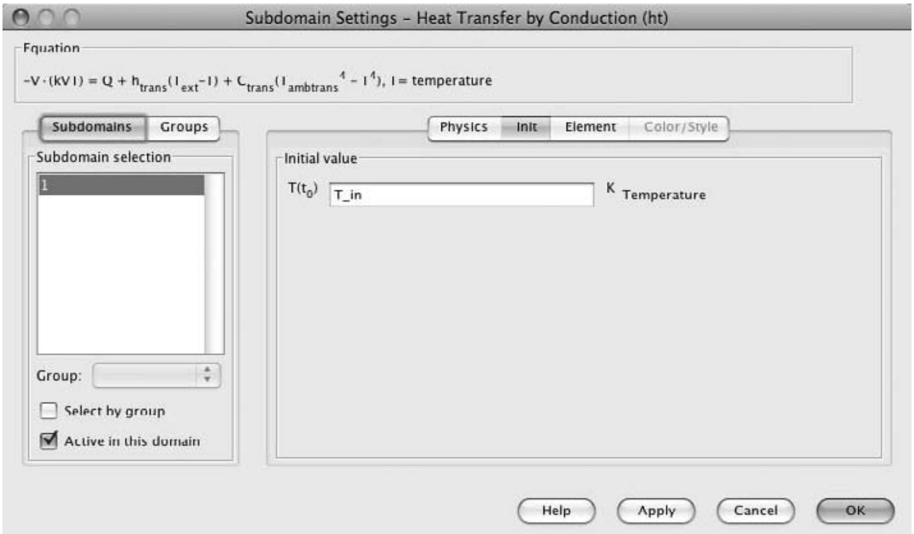


FIGURE 1.10 1D Subdomain Settings Init window settings

Now the modeler needs to enter the appropriate Boundary Settings. Using the menu bar, select Physics > Boundary Settings. After the Boundary Settings window appears, as shown in Figure 1.11, select "1" in the Boundary Settings Boundary selection window.

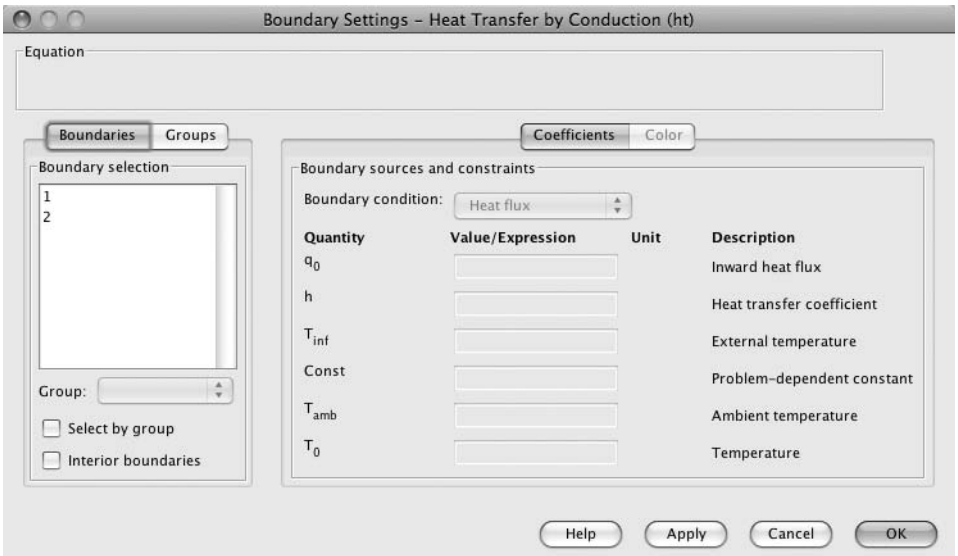


FIGURE 1.11 Boundary Settings window

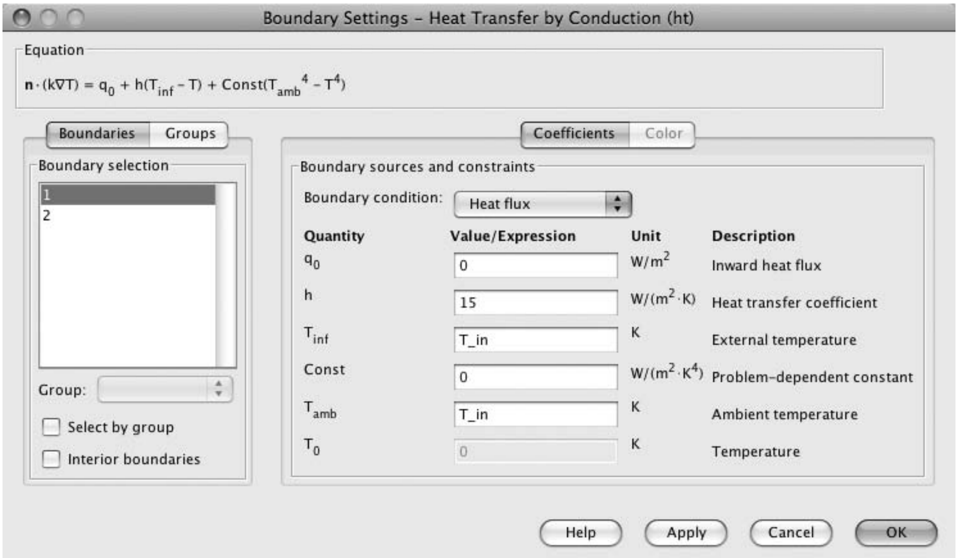


FIGURE 1.12 Filled-in Boundary Settings window for boundary 1

Next, select "Heat flux" from the Boundary conditions pull-down menu. Enter 15 in the Heat transfer coefficient window (h). Enter T_in in the External temperature window (T_{inf}). Enter T_in in the Ambient temperature window (T_{amb}). Click the Apply button. Figure 1.12 shows the filled-in Boundary Settings window for boundary 1.

Now select "2" in the Boundary Settings Boundary selection window. Select "Heat flux" as the Boundary condition. Enter 15 in the Heat transfer coefficient window (h). Enter T_out in the External temperature window (T_{inf}). Enter T_out in the Ambient temperature window (T_{amb}). Click the Apply button. Figure 1.13 shows the filled-in Boundary Settings window for boundary 2. Click OK.

All the Subdomain Settings and the Boundary Settings have now been either chosen or entered. The next step is to mesh the model. In this simple model, all the modeler needs to do is use the toolbar and select "Initialize Mesh." Figure 1.14 shows the initial mesh. The line segments between the dots are the mesh elements.

To improve the resolution, the mesh will be refined twice. All the modeler needs to do is use the toolbar and select Refine Mesh > Refine Mesh. The refined mesh of the single-pane model now contains the 60 elements shown in Figure 1.15, rather than the original 15 elements shown in Figure 1.14.

For this model, the software will be allowed to automatically select the Solver and Solver Parameters. To solve this model, go to the menu bar and select Solve > Solve Problem. The solution is found almost immediately, in 0.014 second. The solution is plotted using the default Postprocessing values and is shown in Figure 1.16.

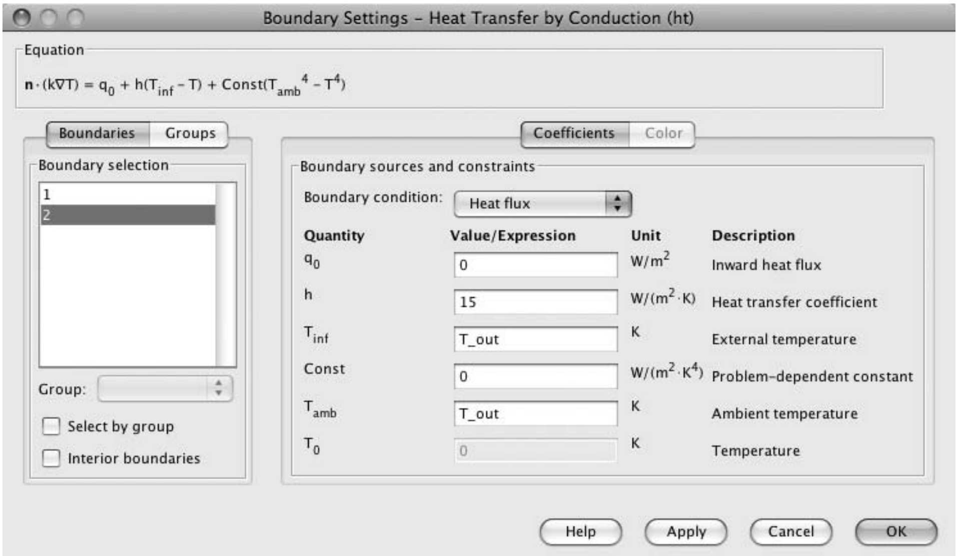


FIGURE 1.13 Filled-in Boundary Settings window for boundary 2

NOTE The precise length of time required for the solution of a given model depends directly on the configuration of the platform and the overhead imposed by the operating system.

NOTE The implicit assumption here would be that the default values in the Postprocessing window are the correct values that the modeler needs to plot the calculated results of the built model. Specifically, that assumption in general will not be true. For example, in the case of this model, the default plot is in Kelvins (K), when the modeler would probably prefer degrees Fahrenheit. Also, it would be helpful to show the change in temperature as a function of the distance into the window pane.

The modeler needs to know before building a new model what the approximate expected resultant values are for the particular properties of the materials selected for use in the model. A firm understanding of the basic physics involved and the appropriate conservation laws that apply to the model are required for analysis, understanding, and configuration of the Postprocessing presentation(s).

The plot presentation is changed as follows: Select Postprocessing > Plot Parameters from the menu bar. When the Plot Parameters window shown in Figure 1.17 appears, select "Line" (Figure 1.18) and then "°F (degF)" from the Unit pull-down bar.

FIGURE 1.14 1D single-pane window with initialized mesh

Select “Use expression to color line.” Click the Color Expression button to display the Line Color Expression window (Figure 1.19). Select “°F (degF)” from the Unit pull-down bar. Click OK, and then click OK again. The Plot Presentation will be rendered as shown in Figure 1.20.

1D Single-Pane Analysis and Conclusions

The 1D single-pane window model, though simple, reveals several fundamental factors about the physics of heat flow through the single-pane window. The interior temperature T_in was established at 70 °F. The exterior temperature T_out was established at 0 °F. The calculated temperature at the midpoint of the single pane is the median value

FIGURE 1.15 1D single-pane window with refined mesh

FIGURE 1.16 1D single-pane window solution plotted using default values

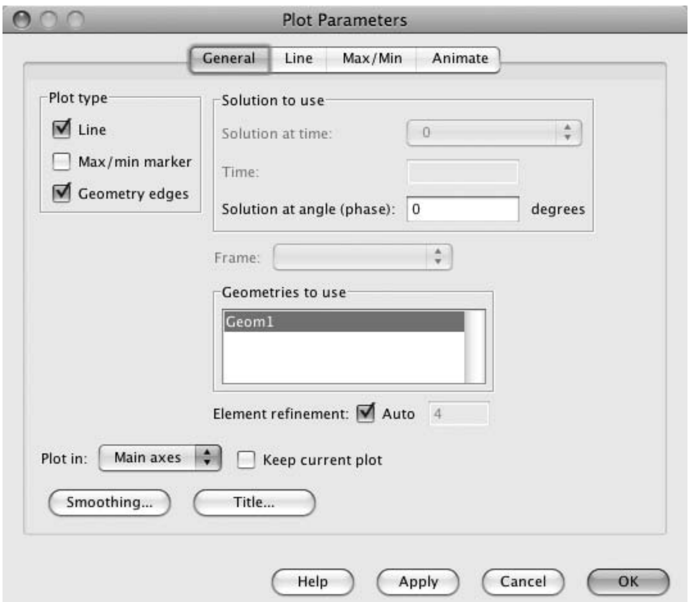


FIGURE 1.17 1D single-pane Plot Parameters General window

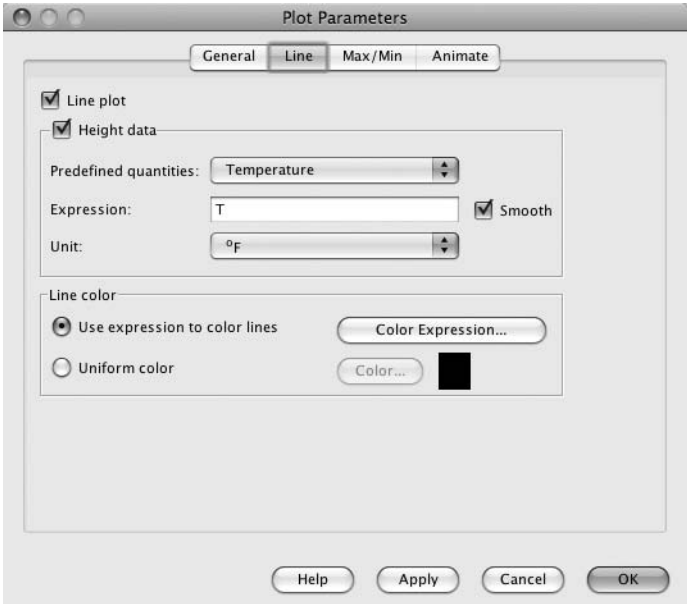


FIGURE 1.18 1D single-pane Plot Parameters Line window

of the interior and exterior temperatures, 35 °F. The temperature difference between the inner surface of the pane and the outer surface of the pane is approximately 2 °F.

The temperature difference between the air in the heated room (70 °F) and the interior surface of the single pane (35.9 °F) is approximately 34 °F. This temperature difference between the ambient temperature and the single-pane window will at least result in water vapor condensation (fogging) and heat loss to the exterior.

NOTE When building models, be sure to *save early and often*.

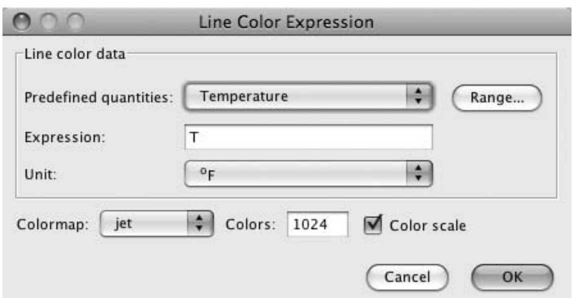


FIGURE 1.19 1D single-pane Line Color Expression window

FIGURE 1.20 1D single-pane window solution plotted using °`F` and Color Bar

1D Dual-Pane Heat Flow Model

This 1D model explores the physics of a dual-pane window with an air space between the panes. This model is parametrically similar to the single-pane window model for ease of comparison of the modeling results.

Run the COMSOL Multiphysics application. Select “New” and then select “1D” in the Model Navigator. Select COMSOL Multiphysics > Heat Transfer > Conduction > Steady-state analysis. Click OK.

After the 1D workspace appears, use the menu bar to select Options > Constants. Import the file ModelOC_1D_WP1.txt saved earlier. To import this file, click on the Folder icon in the lower-left corner of the Constants window. These imported entries, as shown in Figure 1.21, define the Interior Temperature, the Exterior Temperature, and the air pressure for use in this model and the models to follow.

Modeling a dual-pane window requires that three lines be drawn in the workspace window. The drawn lines represent the left (first) pane, the air space, and the right (second) pane, respectively. To use the menu bar to draw the first line, select Draw > Specify Objects > Line. Enter 0.000 space 0.005 in the window, as shown in Figure 1.22. Leave the default Polyline, and click the OK button. Next, use the menu bar to draw the second line. Select Draw > Specify Objects > Line. Enter 0.005 space 0.015 in the window, as shown in Figure 1.23. Leave the default Polyline, and click the OK

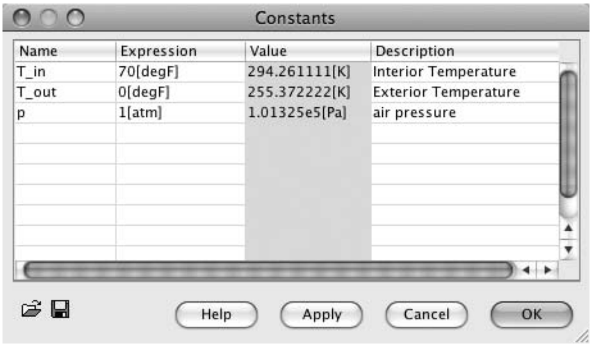


FIGURE 1.21 1D dual-pane Constants specification window

button. Finally, using the menu bar, draw the third line. Select Draw > Specify Objects > Line. Enter 0.015 space 0.020 in the window. Leave the default Polyline, and click the OK button. Figure 1.24 shows the results of the model line creation before clicking the Zoom Extents icon in the toolbar.

Once the Zoom Extents icon is clicked, the specified dual-pane Window model will appear in the workspace as shown in Figure 1.25.

Next, using the menu bar, select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select “1” in the Subdomain selection window. For this model, select Load > Basic Materials Properties > Silica Glass. Click OK. All the appropriate values displayed in the Value/Expression subwindows in the Subdomain Settings window are altered as the new values are loaded from the library. (Silica glass has a thermal conductivity value roughly 0.35% of that of copper.)

Once the thermal conductivity is loaded from the materials properties library, enter T_in in the T_{ext} and $T_{ambtrans}$ windows, as shown in Figure 1.26. Click the Apply button.

Next, select “2” in the Subdomain selection window. For this model, select Load > Basic Materials Properties > Air, 1 atm. Click OK. All the appropriate values displayed in the Value/Expression subwindows in the Subdomain Settings window are altered as the new values are loaded from the library.

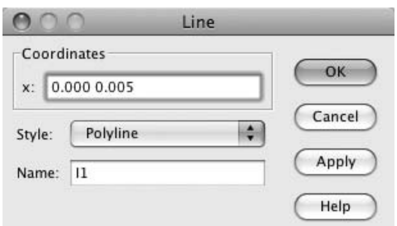


FIGURE 1.22 1D dual-pane Line specification window for the left pane

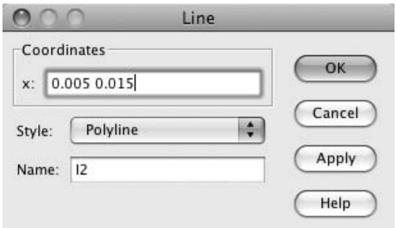


FIGURE 1.23 1D dual-pane Line specification window for the air gap

Once the thermal conductivity is loaded from the materials properties library, enter T_in in the T_{ext} and $T_{ambtrans}$ windows, as shown in Figure 1.27. Click the Apply button.

Next, select “3” in the Subdomain selection window. For this model, Select Library materials list > Silica Glass. The previously selected values loaded from the materials library are loaded into this subdomain.

Once the thermal conductivity is loaded from the materials properties library, enter T_out in the T_{ext} and $T_{ambtrans}$ windows, as shown in Figure 1.28. Click the Apply button.

Next, set the initial conditions for each subdomain (1, 2, 3) by clicking the Init button and then entering the initial conditions shown in Table 1.1. Click OK.

Now, the modeler needs to enter the appropriate Boundary Settings. Using the menu bar, select Physics > Boundary Settings, as shown in Figure 1.29.

FIGURE 1.24 1D dual-pane workspace showing both panes and the air gap

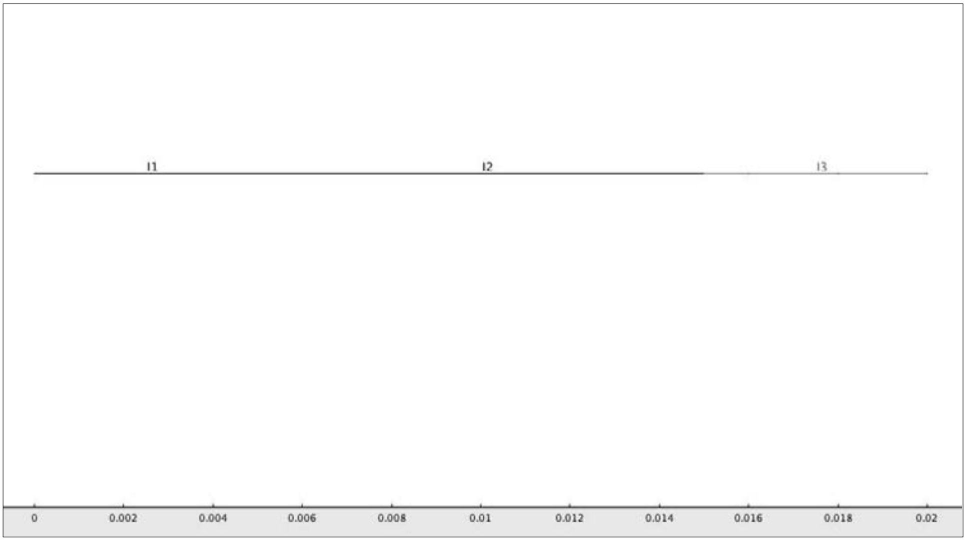


FIGURE 1.25 1D dual-pane workspace after clicking the Zoom Extents icon

Once the Boundary Settings window appears, select "1" in the Boundary selection window. Select "Heat flux" as the Boundary condition. Enter 15 in the Heat transfer coefficient window (h). Enter T_in in the External temperature window (T_{inf}). Enter T_in in the Ambient temperature window (T_{amb}). Click the Apply button. Figure 1.30 shows the filled-in Boundary Settings window for boundary 1.

FIGURE 1.26 1D dual-pane Subdomain Settings Physics window settings, left pane

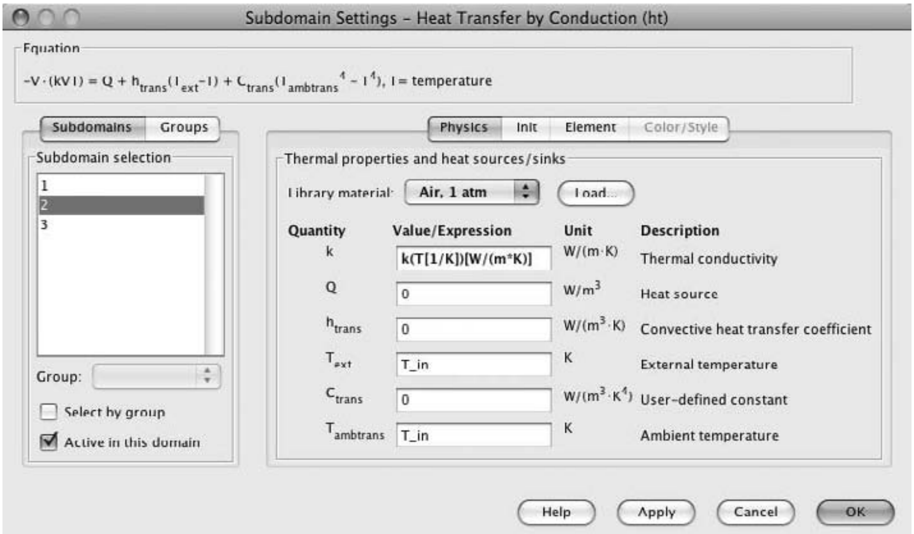


FIGURE 1.27 1D dual-pane Subdomain Settings Physics window settings, air gap

Table 1.1 Subdomain Settings, Initial Conditions

Subdomain	1	2	3
Init	T_in	T_in	T_out

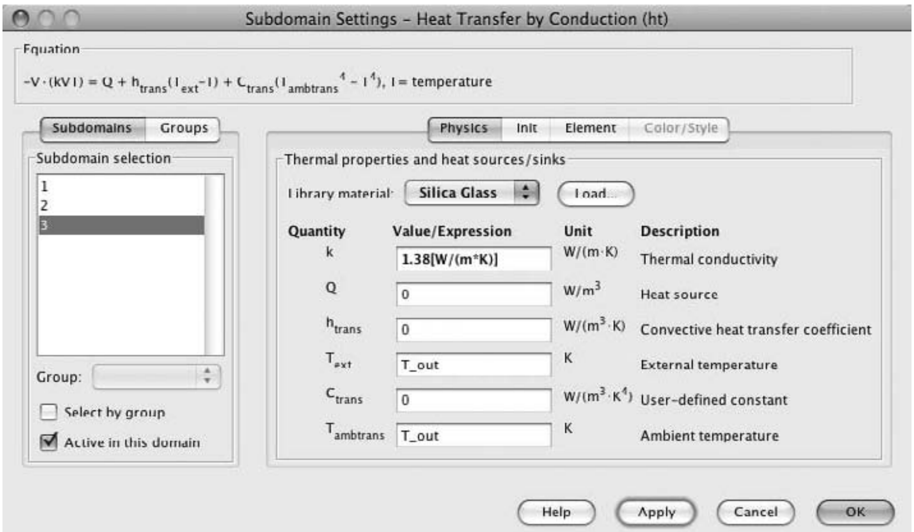


FIGURE 1.28 1D dual-pane Subdomain Settings Physics window settings, right pane

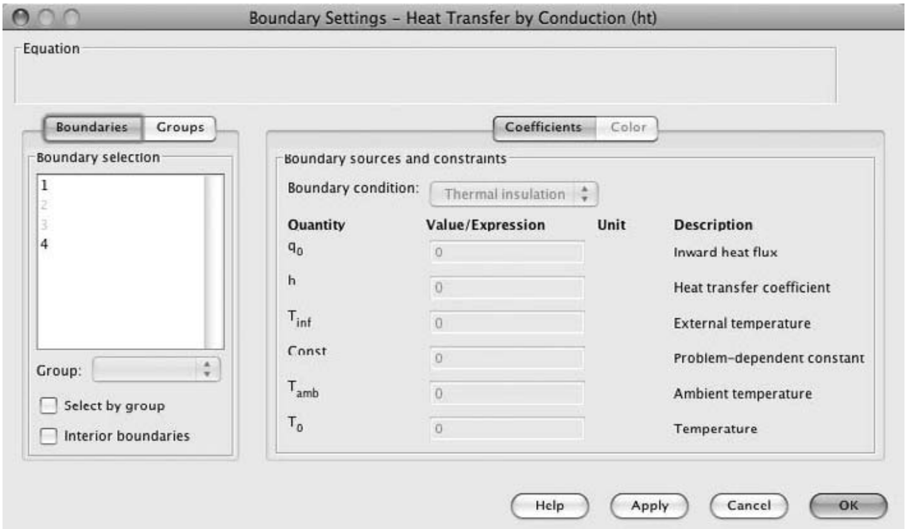


FIGURE 1.29 1D dual-pane Boundary Settings window

Now select "4" in the Boundary Settings Boundary selection window. Select "Heat flux" as the Boundary condition. Enter 15 in the Heat transfer coefficient window (h). Enter T_out in the External temperature window (T_{inf}). Enter T_out in the Ambient temperature window (T_{amb}). Click the Apply button. Figure 1.31 shows the filled-in Boundary Settings window for boundary 4. Click OK.

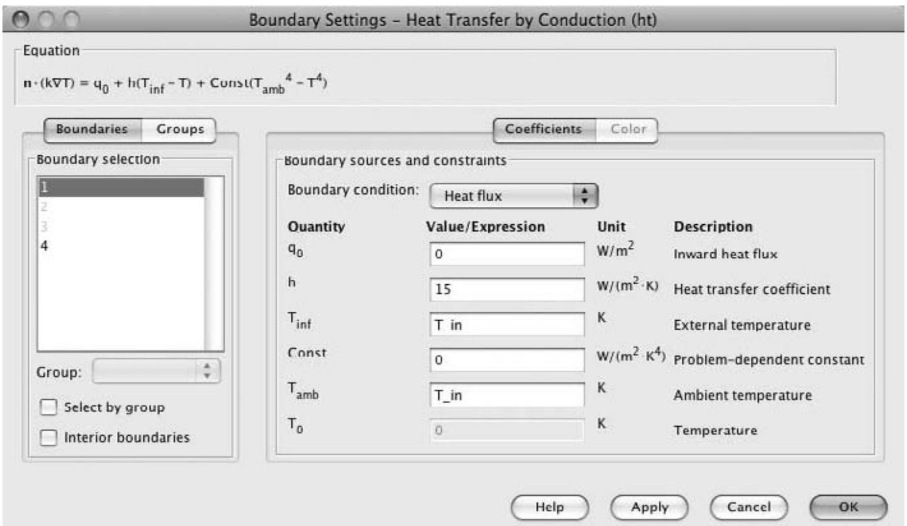


FIGURE 1.30 Filled-in 1D dual-pane Boundary Settings window for boundary 1

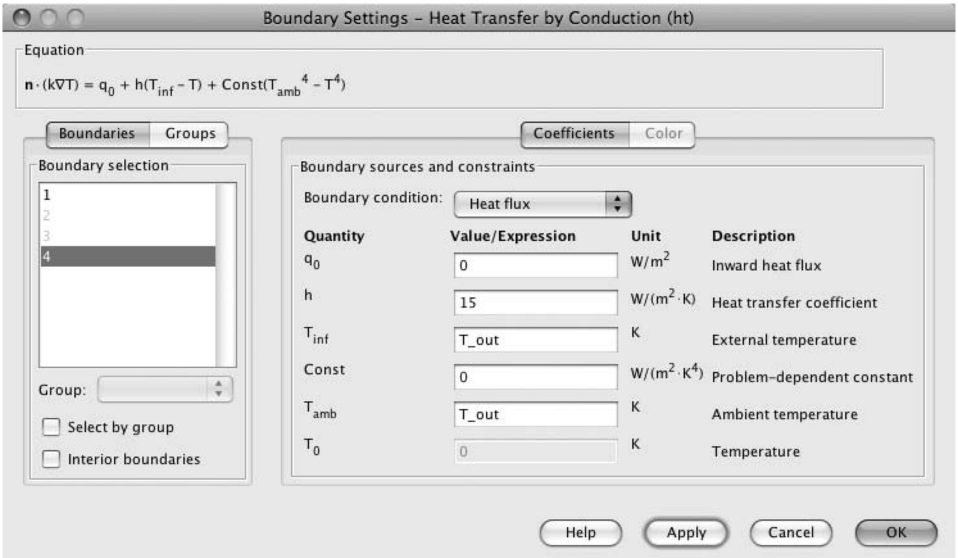


FIGURE 1.31 Filled-in 1D dual-pane Boundary Settings window for boundary 4

NOTE At this point, a new modeler probably wonders why no conditions have been specified for boundaries 2 and 3. The COMSOL Multiphysics software default condition is to automatically establish continuity for interior boundaries. The numbers for boundaries 2 and 3 are grayed out to indicate that they are not available for setting. The default boundary settings can be overridden, if needed, by the advanced modeler by clicking the Interior boundaries check box to make boundaries 2 and 3 accessible.

Once all the Subdomain Settings and the Boundary Settings for this model have been either chosen or entered, the next step is to mesh the model. In this simple model, all the modeler needs to do is use the menu bar and Select "Initialize Mesh." Figure 1.32 shows the initial mesh with 16 elements.

The mesh will be refined twice to improve the resolution. All the modeler needs to do is use the toolbar and select Refine Mesh > Refine Mesh. The refined mesh of the dual-pane model now contains the 64 elements shown in Figure 1.33, rather than the original 16 elements shown in Figure 1.32.

For this model, the software will be allowed to automatically select the Solver and Solver Parameters. To solve this model, go to the menu bar and select Solve > Solve Problem. The solution is found almost immediately, in 0.317 second (the time to solution will vary, depending on the platform). The solution is plotted using the default Postprocessing values and is shown in Figure 1.34.

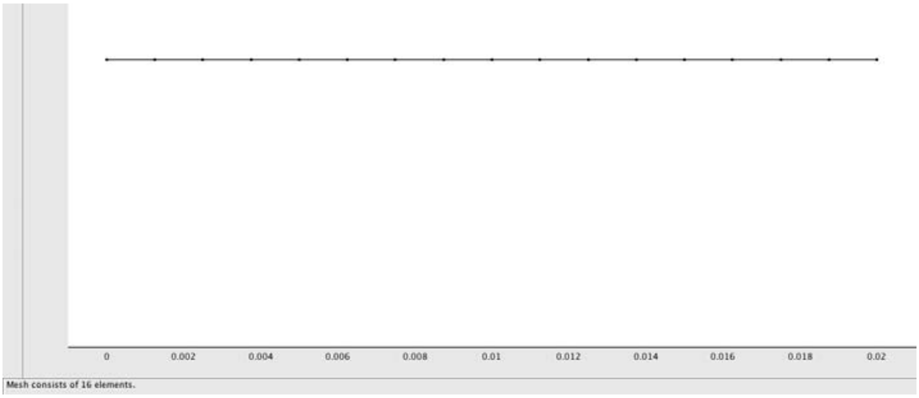


FIGURE 1.32 1D dual-pane window with air gap with initialized mesh

---

NOTE The implicit assumption here would be that the default values in the Postprocessing window are the correct values that the modeler needs to plot the calculated results of the built model. Specifically, that assumption in general will not be true. The modeler needs to know before building a new model what the approximate expected resultant values are for the particular properties of the materials selected for use in the model. A firm understanding of the basic physics involved and the appropriate conservation laws that apply to the model are required for analysis, understanding, and configuration of the Postprocessing presentation(s).

The plot presentation is changed as follows: Select Postprocessing > Plot Parameters from the menu bar. When the Plot Parameters window shown in Figure 1.35 appears, select "Line" (Figure 1.36) and then "°F (degF)" from the Unit pull-down bar (Figure 1.37). Select "Use expression to color lines." Click the Color Expression button

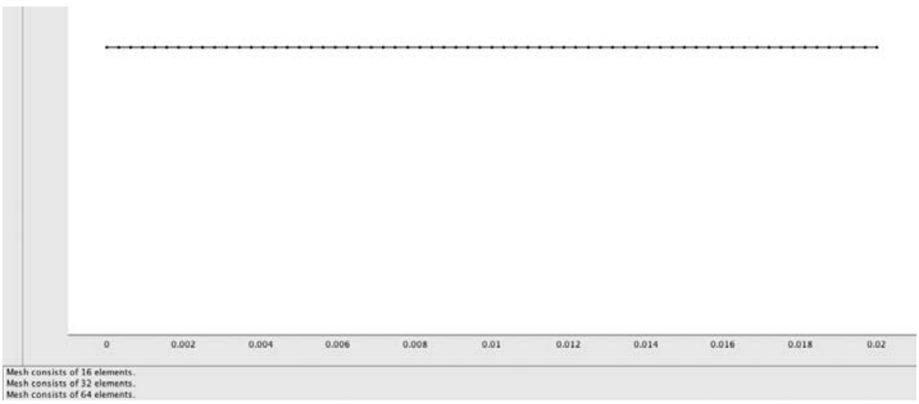


FIGURE 1.33 1D dual-pane window with air gap with refined mesh

FIGURE 1.34 1D dual-pane window solution plotted using default values

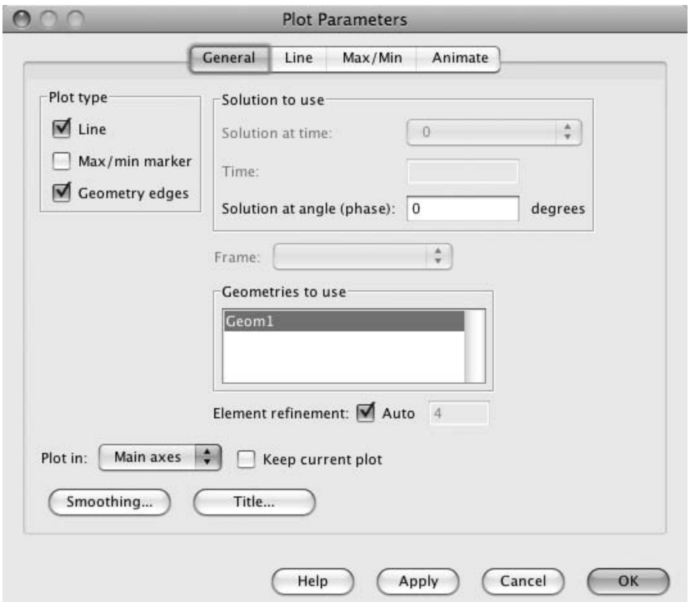


FIGURE 1.35 1D dual-pane Plot Parameters General window

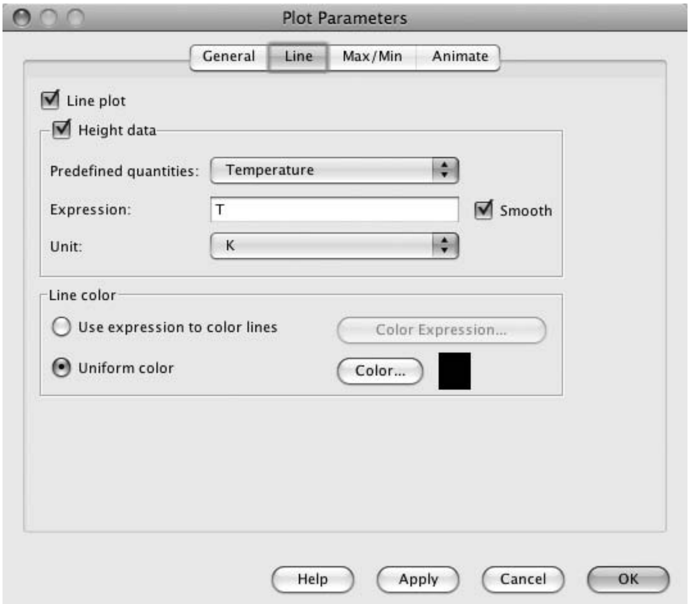


FIGURE 1.36 1D dual-pane Plot Parameters Line window

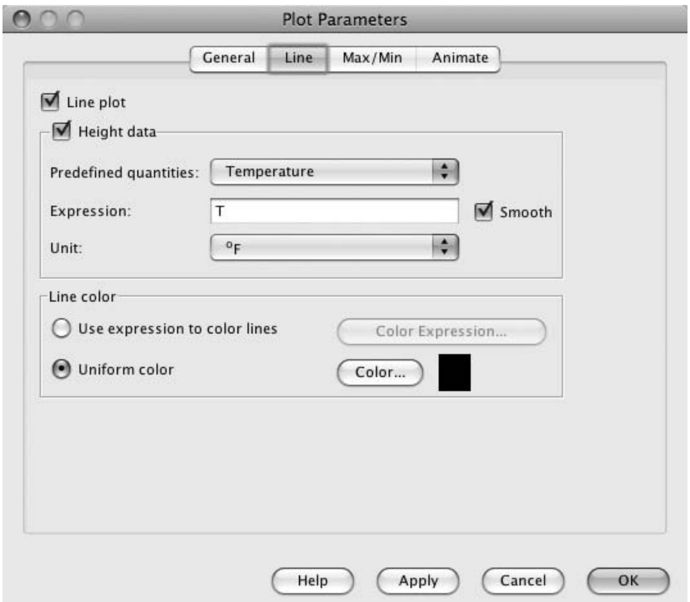


FIGURE 1.37 1D dual-pane window solution set to use °F

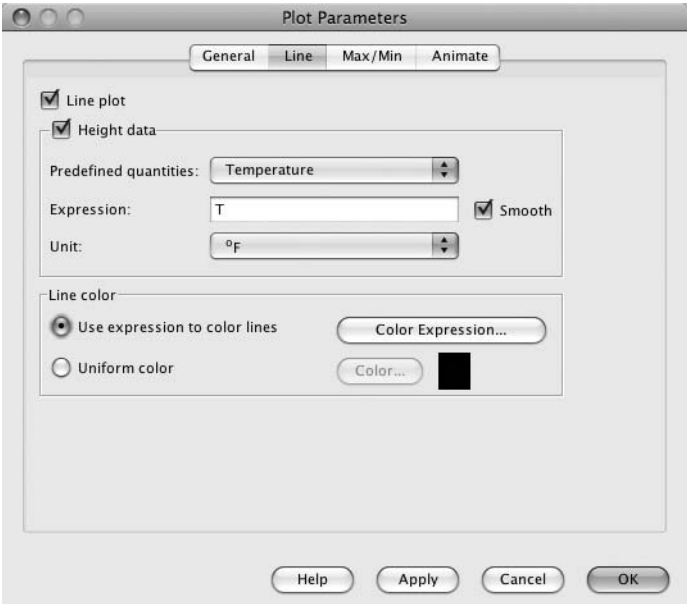


FIGURE 1.38 1D dual-pane window solution set to use Color Expression

(Figure 1.38). Select "°F (degF)" from the Unit pull-down bar (Figure 1.39). Click OK, and then click OK again. The plot presentation will be rendered as shown in Figure 1.40.

Dual-Pane Analysis and Conclusions

The 1D dual-pane window model, though simple, reveals several fundamental factors about the physics of heat flow through the dual-pane window. The interior temperature T_in was established at 70 °F, as in the single-pane model. The exterior temperature T_out was established at 0 °F, as in the single-pane model. The calculated temperature at the midpoint of the single pane is the median value of the interior and

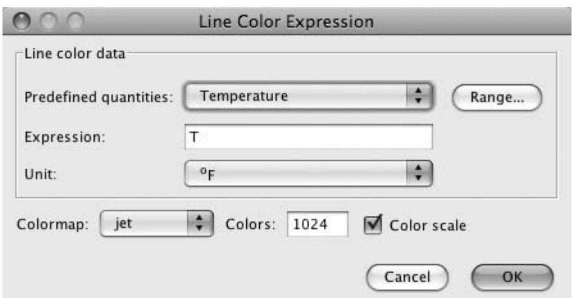


FIGURE 1.39 1D single-pane window solution set to use °F

FIGURE 1.40 1D dual-pane window solution plotted using °F and Color Bar

exterior temperatures, 35 °F. The temperature difference between the inner surface of the left pane and the outer surface of the left pane is approximately 0.48 °F. The temperature difference between the inner surface of the right pane and the outer surface of the right pane is also approximately 0.48 °F. The temperature difference between the inner surface and the outer surface of the dual-pane window is approximately 53 °F, as compared to approximately 2 °F for the single-pane window.

The temperature difference between the air in the heated room (70 °F) and the interior surface (61.5 °F) of the dual-pane window is approximately 8.5 °F. This small temperature difference will result in some heat loss and minimal water vapor condensation (fogging).

Compare the result for the dual-pane window to that of the single-pane window. The temperature difference between the air in the heated room (70 °F) and the interior surface of the single-pane window (35.9 °F) is approximately 34 °F. This temperature difference between the ambient temperature and the single-pane window will at least result in water vapor condensation (fogging) and heat loss to the exterior.

1D Triple-Pane Heat Flow Model

This 1D model explores the physics of a triple-pane window with an air space between each pair of panes. This model is parametrically similar to the single-pane and dual-pane models for ease of comparison of the modeling results.

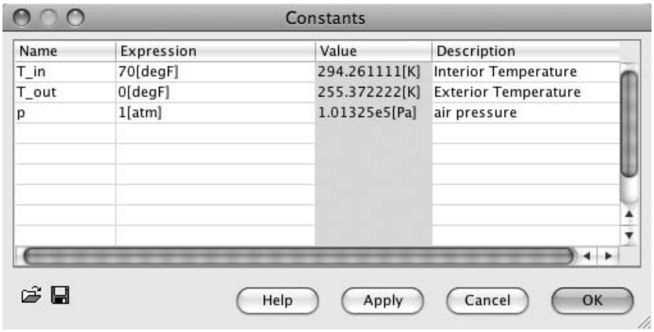


FIGURE 1.41 1D triple-pane Constants specification window

Run the COMSOL Multiphysics application. Select "New" and then "1D" in the Model Navigator. Select COMSOL Multiphysics > Heat Transfer > Conduction > Steady-state analysis. Click the OK button.

After the 1D workspace appears, use the menu bar to select Options > Constants. Import the file ModelOC_1D_WP1.txt saved earlier. To import this file, click on the Folder icon in the lower-left corner of the Constants window. These imported entries, as shown in Figure 1.41, define the Interior Temperature, the Exterior Temperature, and the air pressure for use in this model and the models to follow.

Modeling a triple-pane window requires that five lines be drawn in the workspace window. The drawn lines represent the left (first) pane, the first air space, the center (second) pane, the second air space, and the right (third) pane, respectively. Use the toolbar to draw the first line: Select Draw > Specify Objects > Line. Enter 0.000 space 0.005 in the window. Leave the default Polyline, and then click the OK button. Next, use the menu bar to draw the remaining four lines, as indicated in Table 1.2. Then, click the Zoom Extents icon in the toolbar. The finished workspace configuration is shown in Figure 1.42.

Table 1.2 Triple-Pane Window Workspace Lines

Line	Start	End
1	0.000	0.005
2	0.005	0.015
3	0.015	0.020
4	0.020	0.030
5	0.030	0.035

FIGURE 1.42 1D triple-pane workspace after clicking the Zoom Extents icon

Next, using the menu bar, select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select "1" in the Subdomain selection window. For this model, select Load > Basic Materials Properties > Silica Glass. Click OK (see Figure 1.43). Enter the remaining Subdomain Settings as shown in Table 1.3.

FIGURE 1.43 1D triple-pane Subdomain Settings Physics window settings

Table 1.3 Triple-Pane Window Subdomain Settings

Subdomain	Material	T_{ext}	$T_{ambtrans}$
1	Silica glass	T_in	T_in
2	Air	T_in	T_in
3	Silica glass	T_in	T_in
4	Air	T_in	T_in
5	Silica glass	T_out	T_out

Table 1.4 Subdomain Settings, Initial Conditions

Subdomain	1	2	3	4	5
Init Setting	T_in	T_in	T_in	T_in	T_out

Next, set the initial conditions for each subdomain (1, 2, 3, 4, 5) by clicking the Init button and then entering the initial conditions shown in Table 1.4. See Figure 1.44.

Having configured the Subdomain Settings for this model, the modeler needs to enter the appropriate Boundary Settings. Using the menu bar, select Physics > Boundary Settings. The Boundary Settings window appears, as shown in Figure 1.45.

FIGURE 1.44 1D triple-pane Subdomain Settings Init window settings

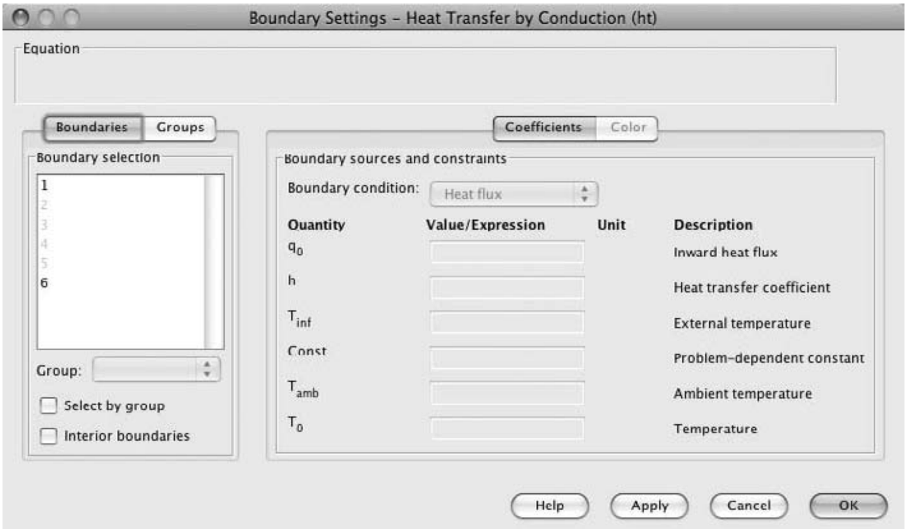


FIGURE 1.45 Blank 1D triple-pane Boundary Settings window

Once the Boundary Settings window appears, select "1" in the Boundary selection window. Select "Heat flux" as the Boundary condition. Enter 15 in the Heat transfer coefficient window (h). Enter T_in in the Ambient temperature window (T_{inf}). Enter T_in in the External temperature window (T_{amb}). Click the Apply button. Figure 1.46 shows the filled-in Boundary Settings window for boundary 1. Fill in the

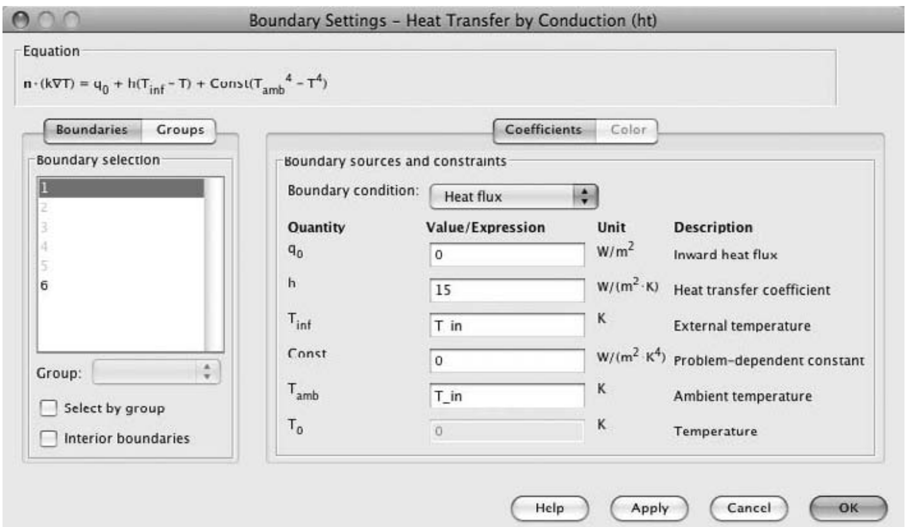


FIGURE 1.46 Filled-in 1D triple-pane Boundary Settings window for boundary 1

Table 1.5 Boundary Settings

Boundary	1	2	3	4	5	6
condition	Heat flux	*	*	*	*	Heat flux
h	15	*	*	*	*	15
T_{inf}	T_in	*	*	*	*	T_out
T_{amb}	T_in	*	*	*	*	T_out

*Do not alter the default setting.

remaining Boundary Settings as shown in Table 1.5. Click the Apply button after each entry, and click OK at the end of the process.

NOTE At this point, as mentioned in the discussion of the dual-pane window model, no conditions are specified for boundaries 2, 3, 4, and 5, because the COMSOL Multiphysics software default condition is to automatically establish continuity for interior boundaries. The numbers for boundaries 2, 3, 4, and 5 are grayed out to indicate that they are not available for setting. The default boundary settings can be overridden, if needed, by the advanced modeler.

Once all the Subdomain Settings and the Boundary Settings for this model have been either chosen or entered, the next step is to mesh the model. In this model, all the modeler needs to do is use the toolbar and select "Initialize Mesh." Figure 1.47 shows the initial mesh with 16 elements.

To improve the resolution, the mesh will be refined twice. All the modeler needs to do is use the menu bar and select Refine Mesh > Refine Mesh. The refined mesh of the triple-pane model now contains the 64 elements shown in Figure 1.48, rather than the original 16 elements shown in Figure 1.47.

FIGURE 1.47 1D triple-pane window with air gaps with initialized mesh

FIGURE 1.48 1D triple-pane window with air gaps with refined mesh

For this model, the software will be allowed to automatically select the Solver and Solver Parameters. To solve this model, go to the menu bar and select Solve > Solve Problem. The solution is found almost immediately, in 0.346 second (the length of time to solution will vary depending on the platform). The solution is plotted using the default Postprocessing values and is shown in Figure 1.49.

FIGURE 1.49 1D triple-pane window solution plotted using default values

FIGURE 1.50 1D triple-pane window solution plotted using °F and Color Bar

---

NOTE As noted earlier in the solutions of the single-pane and dual-pane models, the postprocessing parameters need to be altered to reveal the most information at a glance.

The plot presentation is changed as follows: Select Postprocessing > Plot Parameters from the menu bar. When the Plot Parameters window appears, select “Line” and then “°F (degF)” from the Unit pull-down bar. Select “Use expression to color lines.” Click the Color Expression button. Select “°F (degF)” from the Unit pull-down bar. Click OK and then click OK again. The plot presentation will be rendered as shown in Figure 1.50.

Triple-Pane Analysis and Conclusions

The 1D triple-pane window model reveals several fundamental factors about the physics of heat flow through the triple-pane window. The interior temperature T_in was established at 70 °F, as in the single-pane and dual-pane models. The exterior temperature T_out was established at 0 °F, as in the single-pane and dual-pane models. The temperature difference between the inner surface and the outer surface of the three different window types are compared as shown in Table 1.6.

The temperature difference between the air in the heated room (70 °F) and the interior surface of the triple-pane window (65.2 °F) is approximately 4.8 °F. This

Table 1.6 Comparison of Single-, Dual-, and Triple-Pane Windows

	Single	Dual	Triple
ΔT (°F): across all panes	2	53	60
ΔT (°F): inner pane surface to room ambient temperature	34	8.5	4.8

minimal temperature difference will result in little heat loss and little, if any, water vapor condensation (fogging).

Comparing the results for the three different window configuration models shows that there will be a large reduction in heat loss and annoyance factors (condensation) associated with a change from a single-pane window to a dual-pane window design. The incremental cost of such a design change would typically be less than 100%. However, adding the third pane to the window design reduces the heat loss by only a few percentage points and adds little to the cosmetic enhancement of the design (lack of fogging).

NOTE One of the basic reasons for modeling potential products is to evaluate their relative performance before the actual building of a first experimental physical model. Comparison of these three window models allows such a comparison to easily be made on a "first principles" basis, which will be discussed in the following section of this chapter. That approach is known as the "Model first, build second" approach to engineering design. When the model properly incorporates the fundamental materials properties and design factors, both the time and the cost to develop a fully functional prototype product are significantly reduced.

First Principles Applied to Model Definition

First principles analysis is an analysis whose basis is intimately tied to the fundamental laws of nature. In the case of models described in this book, the modeler should be able to demonstrate both to himself or herself and to others that the calculated results derived from those models are consistent with the laws of physics and the observed properties of materials. Basically, the laws of physics require that what goes in (e.g., as mass, energy, charge) must come out (e.g., as mass, energy, charge) or must accumulate within the boundaries of the model.

NOTE In the COMSOL Multiphysics software, the default interior boundary conditions are set to apply the conditions of continuity in the absence of sources (e.g., heat generation, charge generation, molecule generation) or sinks (e.g., heat loss, charge recombination, molecule loss).

The careful modeler will be able to determine by inspection that the appropriate factors have been considered in the development of the specifications for the various geometries, for the material properties of each subdomain, and for the boundary conditions. He or she must also be knowledgeable of the implicit assumptions and default specifications that are normally incorporated into the COMSOL Multiphysics software model, when a model is built using the default settings.

Consider, for example, the three window models developed earlier in this chapter. By choosing to develop those models in the simplest 1D geometrical mode, the implicit assumption was made that the heat flow occurred in only one direction. That direction was basically normal to the surface of the window and from the high temperature (inside temperature) to the low temperature (outside temperature), as shown by the heat flow indicator in Figure 1.51.

That assumption essentially eliminates the consideration of heat flow along other paths, such as through the window frame, through air leaks around the panes, and so forth. It also assumes that the materials are homogeneous and isotropic, and that there are no thin thermal barriers at the surfaces of the panes. None of these assumptions is typically true in the general case. However, by making such assumptions, it is possible to easily build a first approximation model.

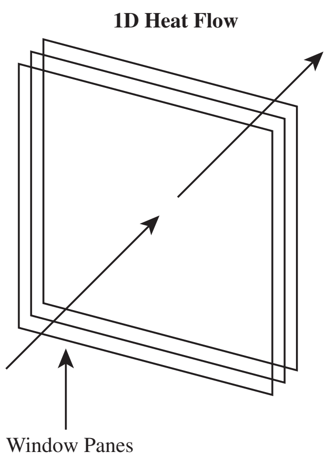


FIGURE 1.51 1D triple-pane window with heat flow indicator

NOTE A first approximation model captures all of the essential features of the problem that needs to be solved, without dwelling excessively on minutiae. A good first approximation model will yield an answer that is sufficiently accurate to enable the modeler to determine whether he or she needs to invest the time and resources necessary to build a higher-dimensionality, significantly more-accurate model.

Common Sources of Modeling Errors

There are four primary sources of modeling errors: insufficient model preparation time, insufficient attention to detail during the model preparation and creation phase, insufficient understanding of the physical and modeling principles required for the creation of an adequate model, and lack of a comprehensive understanding of what defines an adequate model in the modeler's context. The most common modeling errors are those that result from the modeler taking insufficient care in either the development of model details or the incorporation of conceptual errors and/or the generation of keying errors during data/parameter/formula entry.

NOTE One primary source of errors occurs during the process of naming variables. The modeler should be careful to *never give the same name to his or her variables as COMSOL gives to the default variables*. COMSOL Multiphysics software seeks a value for the designated variable everywhere within its operating domain. If two or more variables have the same designation, an error is created. Also, it is best to avoid human errors by using uniquely distinguishable characters in variable names (i.e., avoid using the lowercase "L," the number "1," and the uppercase "I," which in some fonts are relatively indistinguishable; similarly, avoid the uppercase "O" and the number zero "0"). Give your variables meaningful names (e.g., T_in, T_out, T_hot). Also, variable names are case sensitive; that is, T_in is not the same as T_IN.

The first rule in model development is to define the nature of the problem to be solved and to specify in detail which aspects of the problem the model will address. The definition of the nature of the problem should include a hierarchical list of the magnitude of the relative contribution of physical properties vital to the functioning of the anticipated model and their relative degree of interaction.

NOTE Examples of typical physical properties that are probably coupled in any developed model are heat and geometrical expansion/contraction (liquid, gas, solid), current flow and heat generation/reduction, phase change and geometrical expansion/contraction (liquid, gas, solid) and/or heat generation/reduction, and chemical reactions. Be sure to investigate your problem and build your model carefully.

Having built the hierarchical list, the modeler should then estimate the best physical, least-coupled, lowest-dimensionality modeling approach to achieve the most meaningful first approximation model.

Exercises

1. Build, mesh, and solve the 1D single-pane window problem presented earlier in this chapter.
2. Build, mesh, and solve the 1D dual-pane window problem presented earlier in this chapter.
3. Build, mesh, and solve the 1D triple-pane window problem presented earlier in this chapter.
4. Add a fourth pane, and build, mesh, and solve the problem. Analyze, compare, and contrast the results with the results of Exercises 1, 2, and 3.

2 Materials and Databases

In This Chapter

■ Materials and Database Guidelines and Considerations

Materials selection and definition are the most important tasks performed by the modeler during the preliminary stages of model building preparation. The selection of appropriate materials is vital to the ultimate functionality of the device or process being modeled. Once the modeler has decided on a good first approximation to the device/process being modeled, the materials selection process begins.

NOTE A good first approximation is a problem statement that incorporates all the essential (first-order) physical properties and functionality of the device/process to be modeled.

Not all properties of all materials are or can be incorporated into the modeling process at the same time, because modeling resources (e.g., computer memory, computer speed, number of cores) are limited. It is important that the modeler start the modeling process by building a model that incorporates the most critical physical and functional aspects of the developmental problem under consideration.

To put the problem in perspective, a simple search of the Web on the term "materials properties database" yields approximately 48 million hits. Obviously, the modeler is not going to exhaustively explore all such links.

NOTE Exploration of any given subset of the 48 million links for the properties of specific materials will reveal several possible standard results: (1) those links do not have a value for the desired material property; (2) those links do have values for the desired material property, but in unconventional units that need to be converted and then

compared for relative accuracy and reliability; (3) some of those links do have values for the desired material property, in the desired units, that need to be compared for relative accuracy and reliability; (4) a link is found that has some of the desired properties for a particular material, but not all of the needed properties; and (5) once found, property values need to be hand-copied and hand-entered into the model. In any case, many hours can be spent trying to obtain and determine accurate values for specific properties of particular materials.

COMSOL® Multiphysics® software and the associated add-on modules include basic materials libraries. The information contained in those basic materials libraries may easily be enhanced by the addition of other materials properties data through several different means. This chapter discusses three solutions to the obtaining and supplying of materials properties values directly to COMSOL Multiphysics software models. Each of these three solutions approaches the problem solution from a different viewpoint, with the same desired result—that is, supplying the modeler with the best materials properties values available to meet the modeler's needs.

■ COMSOL® Material Library Module: Searchable Materials Library

The COMSOL Material Library[1] is a module that can be added through licensing to the basic COMSOL Multiphysics software package to expand the included basic library. The COMSOL Material Library Module has data on approximately 2500 materials including elements, minerals, soil, metal alloys, oxides, steels, thermal insulators, semiconductors, and optical materials, at the minimum. It is searchable by name, DIN,[2] and UNS[3] numbers. Each material can have a maximum of 27 defined properties. Each of those defined properties is available as a function of temperature.

There are two methods to gain access to the Material Library. The first method is through the Options menu. This path can be used to screen materials in advance of building a model. The second method is through the Load button on the Subdomain Settings page. This path incorporates the materials into the model library. Once the Material Library Module has been activated, the technique for using the library is the same. In this example, the Options menu route is used:

1. Activate the COMSOL Multiphysics application.
2. Select COMSOL Multiphysics > Heat Transfer > Convection and Conduction > Steady-state analysis. See Figure 2.1.
3. Click OK.
4. Select Options > Materials/Coefficients Library. See Figure 2.2.

The modeler can determine if the items in the Material Library Module are available by viewing the list in the Materials selection window. The first entry in the

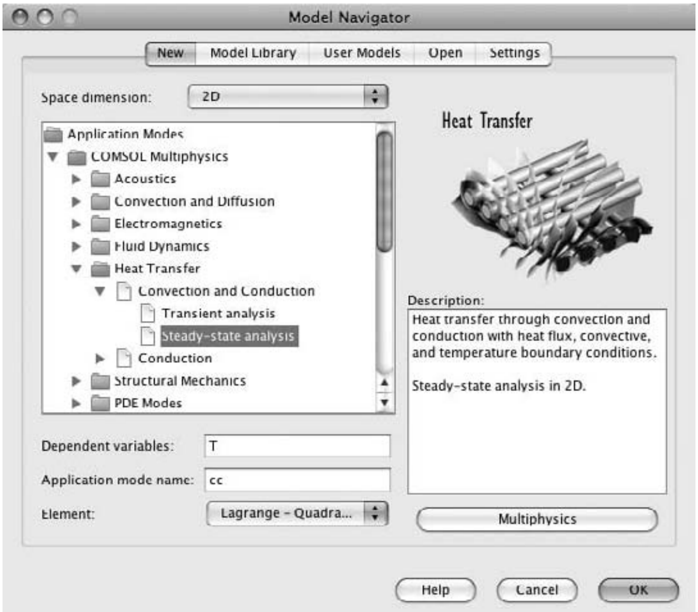


FIGURE 2.1 Model Navigator window

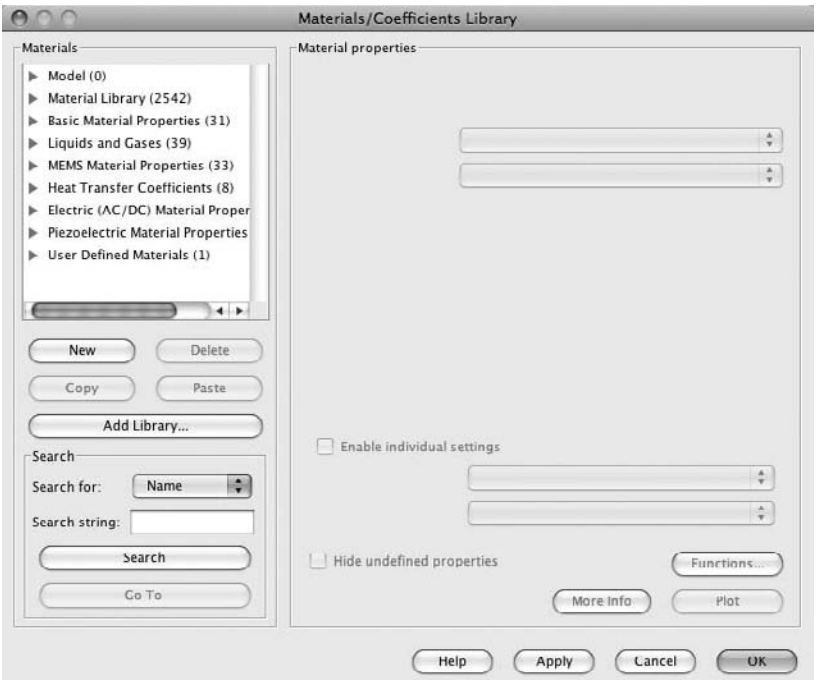


FIGURE 2.2 Materials/Coefficients Library search and/or selection window

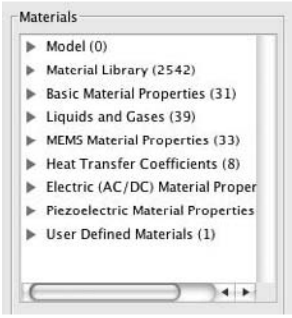


FIGURE 2.3 Materials/Coefficients Library Materials selection window

Materials selection list is Model (0), which indicates that none of the materials in any of the libraries have been selected for use in the current model. The second entry in the Materials selection list is Material Library (2542); it indicates that 2542 materials are available to be selected for use in the current model. See Figure 2.3.

Suppose, for example, the modeler is interested in the properties of copper and copper alloys. He or she would follow these steps:

1. Enter "copper" in the Search string window.
2. Click the Search button. The search results show 87 possible materials are in the library. See Figure 2.4.

FIGURE 2.4 Materials/Coefficients Library search results

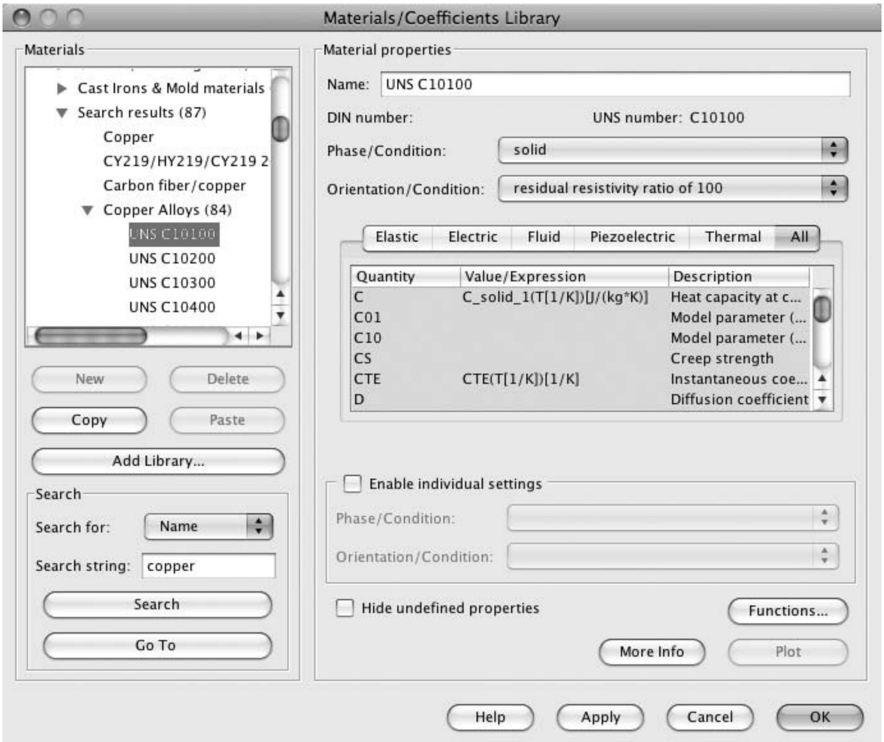


FIGURE 2.5 Materials/Coefficients Library, UNS C10100 Material properties window

3. Select Copper Alloys (84) > UNS C10100. The UNS C10100 is what is known as oxygen-free copper.[4] This high-quality copper is widely used in the electronics industry. The properties of UNS C10100 are shown in the Material properties display window. See Figure 2.5.
4. To see only the defined properties, check the Hide undefined properties check box. See Figures 2.6 and 2.7.

   A similar process can be followed for other material choices.

MatWeb: Searchable Materials Properties Website

MatWeb[5] is an online searchable subscription materials properties data source. MatWeb has 69,000 data sheets for materials that include plastics, metals, ceramics, semiconductors, fibers, and various other commercially available materials. See Figure 2.8.

MatWeb has three classes of access: Un-Registered (free limited feature access), Registered Member (free expanded feature access), and Premium Member (fee-based access to all features, plus selected data storage and modeling software formatted data

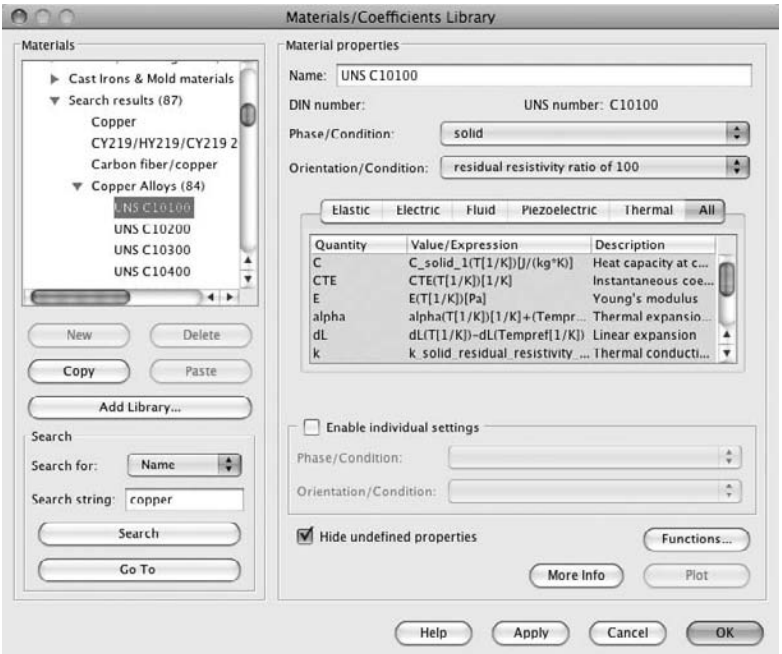


FIGURE 2.6 Materials/Coefficients Library, UNS C10100 defined properties, first half

FIGURE 2.7 Materials/Coefficients Library, UNS C10100 defined properties, second half

MatWeb, Your Source for Materials Information

What is MatWeb? MatWeb's **searchable database of material properties** includes data sheets of thermoplastic and thermoset polymers such as ABS, nylon, polycarbonate, polyester, polyethylene and polypropylene; metals such as aluminum, cobalt, copper, lead, magnesium, nickel, steel, superalloys, titanium and zinc alloys; ceramics; plus semiconductors, fibers, and other engineering materials.

FIGURE 2.8 MatWeb site, home page

MatWeb Feature	Un-Registered	Registered	Premium
Material Data Sheets	√	√	√
- View Any of MatWeb's 69,000 Data Sheets For FREE			
Basic Search Engines	√	√	√
- Key Word or Phrase - Metal Composition - Manufacturer *- Property, Metric or English Units - Material Type - Trade Name*	Up to 1000 results	Up to 1500 results	Up to 2000 results
Advanced Search Engine		√	√
- Combines Text, Category, Property, and Composition Searches		Search on up to 3 criteria at a time	Search on up to 10 criteria at a time
View Property Data in Search Results	√	√	√
Sort Search Results			√
- Sort by Material Name and Property Data in Search Results *- Ascending Value, Descending Value, or Alphabetical by Name*			Sort on any of first 3 numerical criteria
Saved Material Folders		√	√
- Organize Your Most-Referenced Materials *- Add, Edit, and Delete Material Folders*		Save up to 1 Material Folder	Save up to 10 Material Folders
Compare Folder Materials		√	√
- Side-By-Side Comparison of Material Property Data *- Graph Property Value vs Material for any Property*		3 Data Sheets per Folder	20 Data Sheets per Folder
Exclude Discontinued Materials		√	√
- Streamline Search Results by Omitting Discontinued Data Sheets			
Export Material Folder Data			√
- Export Materials to CSV/Excel format. *- Export Data Sheets to a SolidWorks/COSMOSWorks Library* *- Export Data Sheets to the ALGOR Library Format* *- Export Data Sheets to a NEiWorks Library* *- Export Data Sheets in the ANSYS Format* *- Export Data Sheets in the Comsol Format* *- Export Data Sheets in the PlassoTech 3G.Author Format*			
Basic Tools	√	√	√
- Unit Converter With Over 150 Units of Measure *- Basic Weight Calculator*			
Advanced Tools		√	√
- Hardness Converter - Moment of Inertia Calculator *- Searchable Glossary - Advanced Weight Calculator*			

FIGURE 2.9 MatWeb membership level features comparison page

export). All features of the following example can be run (for free) as a Registered user, except for the export feature. To export the selected data, the modeler needs to acquire a Premium membership. See Figure 2.9.

The following example shows the results of a Premium membership search.

1. After login, select "Metal UNS Number" on the login home page. See Figure 2.10.
2. The Web page is shifted to the Metal Alloy UNS Number Search page. See Figure 2.11.

FIGURE 2.10 MatWeb selection search types, login home page

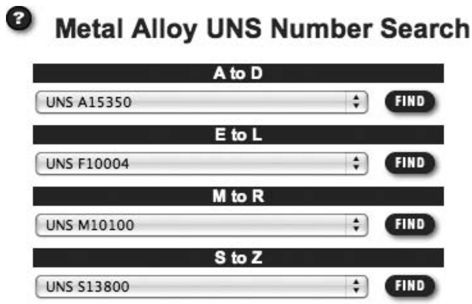


FIGURE 2.11 MatWeb Metal Alloy UNS Number Search selection page

3. Select “UNS C10100” from the A to D drop-down list. See Figure 2.12.
4. Click the FIND button to the right of the selected material number. The search has found 22 data sheets for UNS C10100 (oxygen-free electronic-grade copper); see Figure 2.13.
5. Using the Task pull-down list in the menu bar, create a folder named Copper.
6. Select item 1. See Figure 2.14.
7. Select “Export to COMSOL” from the task list in the menu bar. The available properties values for UNS C10100 are exported as a text file to the modeler’s computer. See Figure 2.15.

   The exported file can be directly imported into COMSOL Multiphysics as follows:

1. Open COMSOL Multiphysics in the application mode of choice.
2. Using the menu bar, select Options > Materials/Coefficients Library.
3. Click the Add Library button.

There are many other features available to the Premium Member at the MatWeb website. Those features can be explored at the modeler’s convenience.

FIGURE 2.12 MatWeb Metal Alloy UNS C10100 selection

FIGURE 2.13 MatWeb search results for UNS C10100 (oxygen-free electronic-grade copper)

FIGURE 2.14 MatWeb selection of UNS C10100 (oxygen-free electronic-grade copper)

FIGURE 2.15 MatWeb properties of UNS C10100 (oxygen-free electronic-grade copper)

PKS-MPD: Searchable Materials Properties Database

PKS-MPD (Pryor Knowledge Systems–Materials Properties Database)[6,7] is a searchable materials properties database with data on more than 4000 materials, including elements, minerals, soil, metals, metal alloys, oxides, steels, thermal insulators, semiconductors, optical materials, and biomaterials (tissue). Each material can have a maximum of 43 defined properties. Each of those defined properties is associated with the temperature of measurement and the frequency of measurement, as available. The collection of defined properties for each materials property datum is exportable in a format suitable for the COMSOL Multiphysics software.

The PKS-MPD selection page, on first use, requires that the modeler choose the version of COMSOL Multiphysics software in use to correctly format the export files (the COMSOL Multiphysics version selection choice remains as chosen until later changed). See Figure 2.16.

Using the same example material as previously, UNS C10100 (oxygen-free electronic-grade copper), the selection criteria can be entered by at least two different paths. To use the first path:

1. Click the Composition tab and select Copper (Cu) from the pull-down list. See Figure 2.17.
2. Click the Add to Search button. See Figure 2.18.

The Selection Criteria window shows that the search yields 440 possible copper-containing candidate materials. Because oxygen-free electronic-grade copper is known to be very pure, the search can be narrowed by adding a specification of the compositional percentage of Cu to the search.

1. Select Copper (Cu) from the element pull-down list.
2. Check the Specify percentage range check box.
3. Enter Min. = 99.9 and Max. = 100 in the appropriate edit windows. See Figure 2.19.
4. Click the Add to Search button. See Figure 2.20.

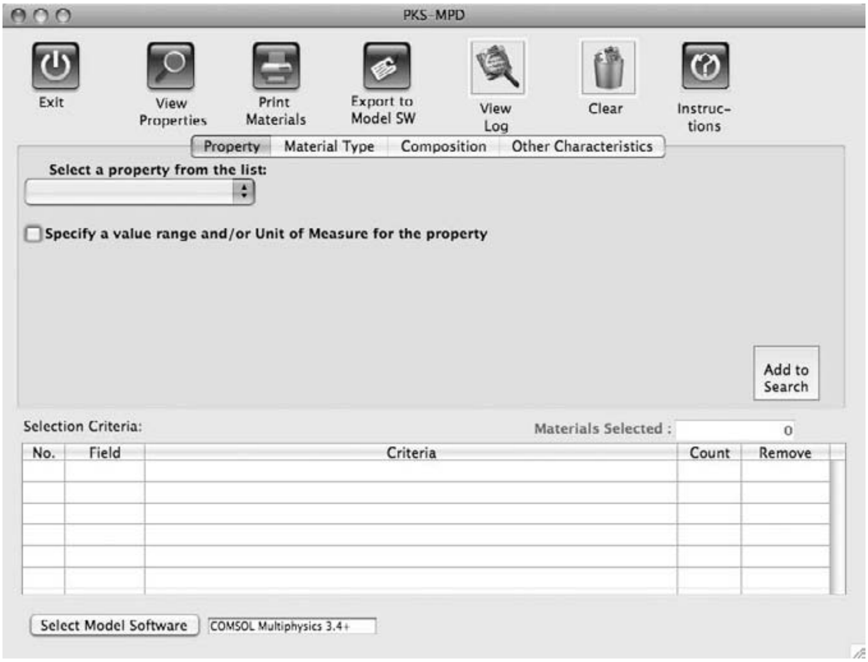


FIGURE 2.16 PKS-MPD main selection page

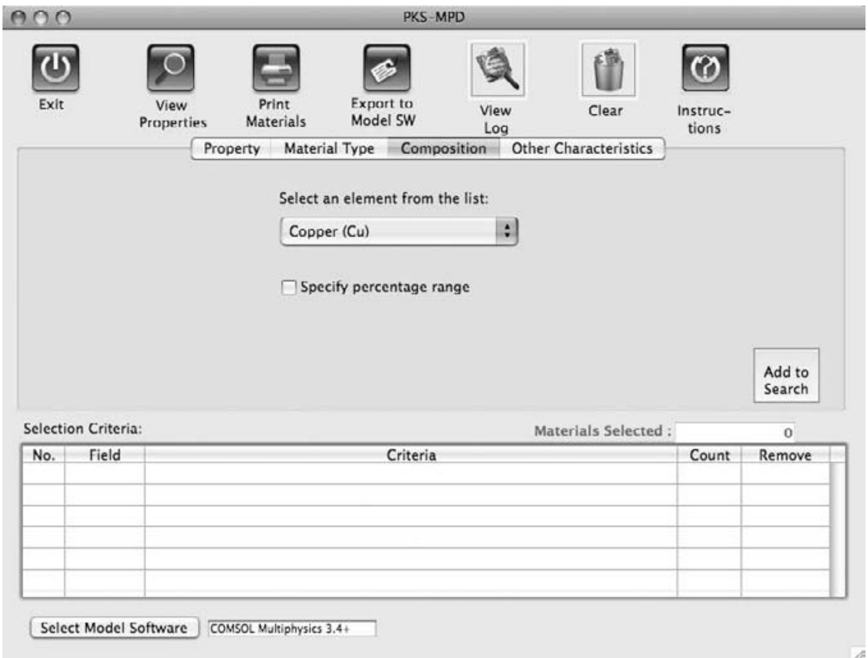


FIGURE 2.17 PKS-MPD Composition selection page for Copper (Cu)

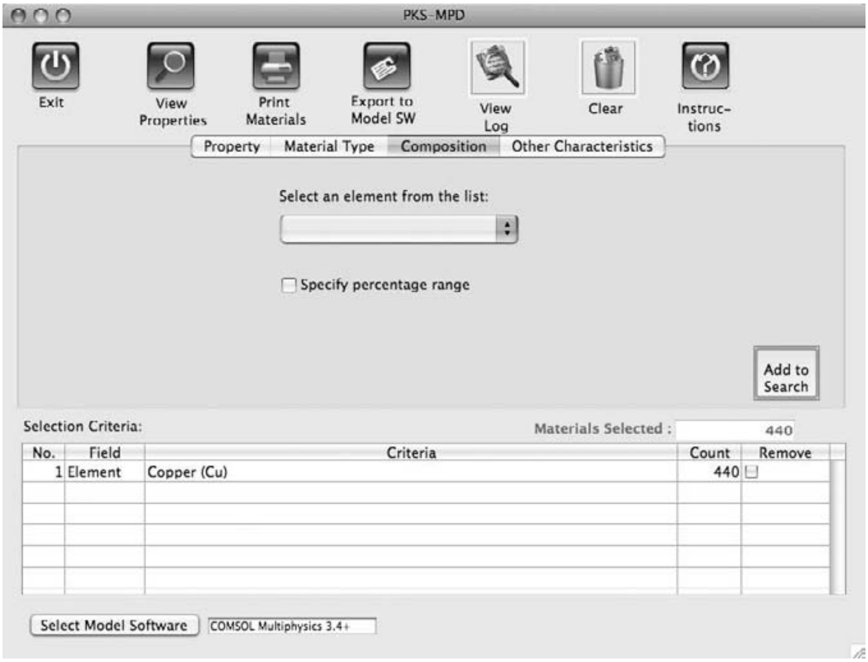


FIGURE 2.18 PKS-MPD Composition selection added page for Copper (Cu)

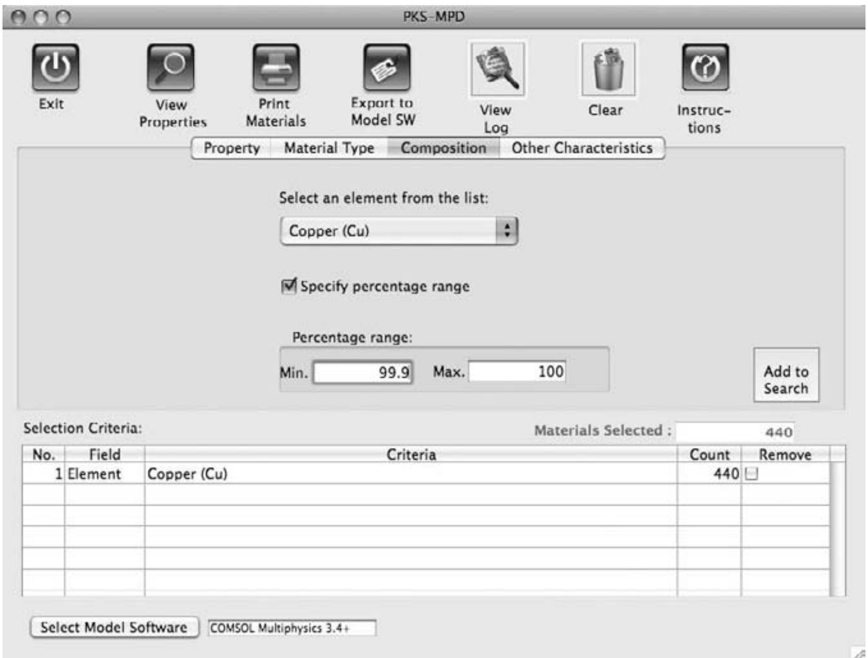


FIGURE 2.19 PKS-MPD Composition percentage range selection page for Copper (Cu)

FIGURE 2.20 PKS-MPD Composition percentage range selection added page for Copper (Cu)

The Selection Criteria window shows that the search yields three possible copper candidate materials. Click the Print Materials button to view the candidate materials and optionally print a data sheet. See Figure 2.21.

The remaining materials candidates are Copper; Copper (UNS C10100); and Copper Alloy, pure copper, UNS C10200.

1. Select “Copper (UNS C10100).” See Figure 2.22.
2. Double-click the selection to view the properties data for the candidate material(s). See Figure 2.23.
3. Click the Accept button (far right, check-marked button).
4. Close the selection window.
5. Click OK on the Page Setup window.
6. Select Print Preview. See Figures 2.24 and 2.25.
7. Click either the Cancel button or the Print button.
8. Click the Export to Model SW button.

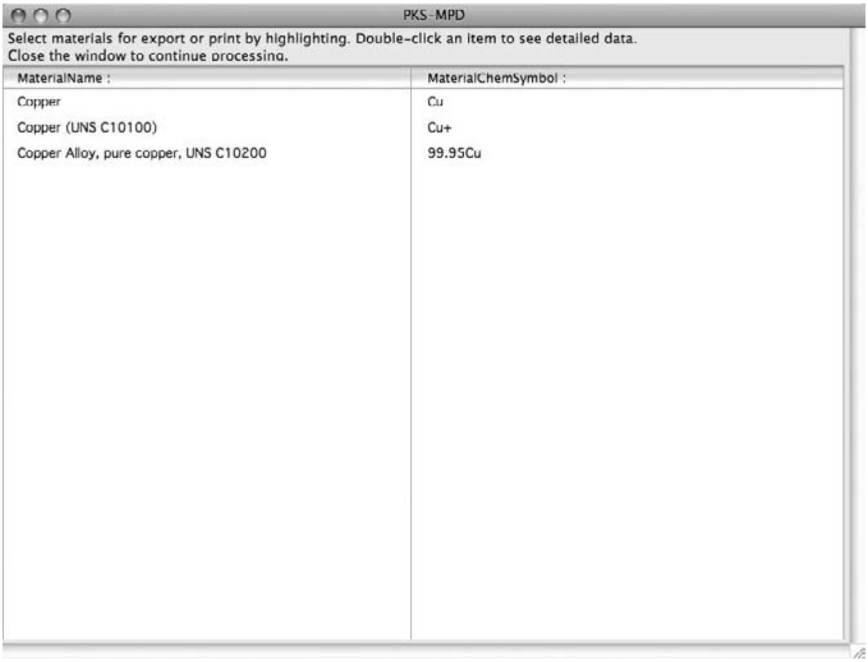


FIGURE 2.21 PKS-MPD Composition materials selection page for Copper (Cu)

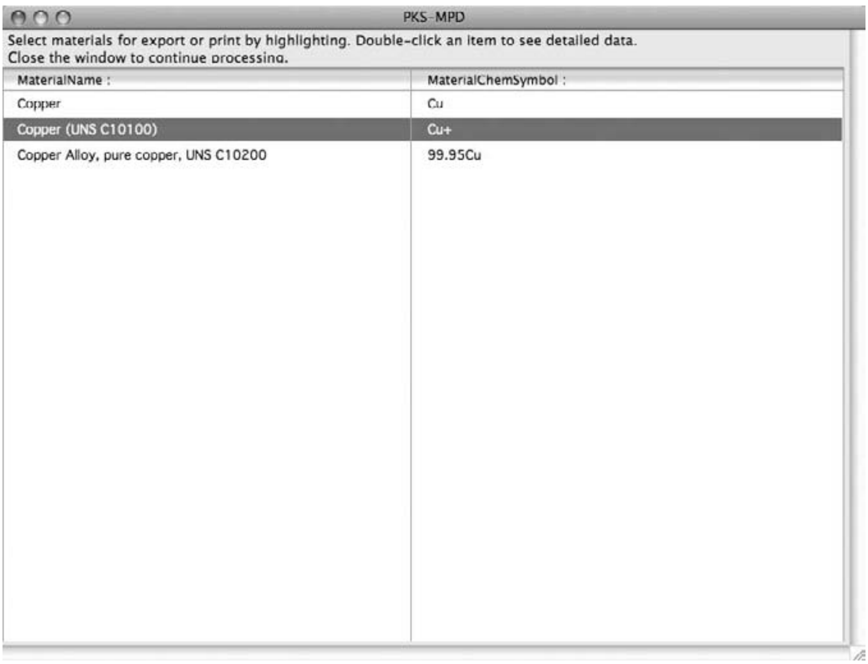


FIGURE 2.22 PKS-MPD Print Materials selection page for Copper (UNS C10100)

FIGURE 2.23 PKS-MPD Materials selection Properties display page for Copper (UNS C10100)

9. Select "Copper (UNS C10100)." See Figure 2.26.
10. Double-click "Copper (UNS C10100)" to verify the candidate material selection choice. See Figure 2.27.
11. Click the Accept button (far right, check-marked button).
12. Close the selection window. See Figure 2.28.
13. Click Yes.
14. Enter Copper (UNS C10100) in the Material Library Name Request window (to provide a name for the entry in the export log).
15. Click OK. See Figure 2.29.

NOTE In the case where the material property is measured under different conditions (e.g., temperature, frequency), the modeler must choose which value he or she wishes to export for use with the material.

Material Properties 6/5/09

Material Name : Copper (UNS C10100)

Formula : Cu+

CAS Reg. No. :

Property / Comments	Symbol/T0 Min	Value/T0 Max	UOM / T0 UOM
Thermal expansion coeff. (alpha) At 20-300 degC (68-570 degF) for ceramic-to-metal seals.	alpha 293.15	1.7700e-5 573.15	1/K K
Yield Strength At 20-300 degC (68-570 degF) for ceramic-to-metal seals.	Ys_pks 293.15	6.9000e+7 573.15	Pa K
Tensile Strength (Syt) At 20-300 degC (68-570 degF) for ceramic-to-metal seals.	Syt 293.15	2.2000e+8 573.15	Pa K
Elongation modulus At 20-300 degC (68-570 degF) for ceramic-to-metal seals.	293.15	45.0 573.15	% K
Young's modulus (E) At 20-300 degC (68-570 degF) for ceramic-to-metal seals.	E 293.15	1.1500e+11 573.15	Pa K
Density (rho) Density may depend considerably on previous treatment.	rho ...	8,960.0 ...	kg/m^3
Boiling Point	bpT_pks	2,868.15	K
Heat Capacity (C)	C	380.0	J/(kg*K)
Heat of fusion	lh_pks	2.1185e+5	J/kg
Thermal expansion coeff. (alpha)	alpha	1.6500e-5	1/K
Thermal conductivity (k) +/- 0.005 cal/cm^2/cm/s/K	k ...	393.9779 ...	W/(m*K)
Electrical Resistivity (res) At 20 degC (68 degF).	res 293.15	1.6730e-8 293.15	ohm-m K
Electrical Conductivity (sigma) At 20 degC (68 degF). Derived from electrical resistivity.	sigma 293.15	5.9773e+7 293.15	S/m K
Young's modulus (E)	E	1.1032e+11	Pa
Melting Point +/- 0.1 degC.	mpT_pks ...	1,356.15 ...	K
Specific gravity Derived from density in g/cm^3.	...	8.96 ...	NONE
Young's modulus (E) At room temperature 20 degC (68 degF).	E 293.15	1.2800e+11 293.15	Pa K
Shear modulus (Gxy) At room temperature 20 degC (68 degF).	Gxy 293.15	4.6800e+10 293.15	Pa K
Poisson's Ratio (nu) At room temperature 20 degC (68 degF).	nu 293.15	3.0800e-1 293.15	NONE K
Yield Strength At room temperature 20 degC (68 degF).	Ys_pks 293.15	3.3300e+7 293.15	Pa K

Page 1

FIGURE 2.24 PKS-MPD Material Properties Print Preview Page 1 for Copper (UNS C10100)

Material Properties 6/5/09

Property / Comments	Symbol/T0 Min	Value/T0 Max	UOM / T0 UOM
Tensile Strength (Syt) At room temperature 20 degC (68 degF).	Syt 293.15	2.0900e+8 293.15	Pa K
Elongation At room temperature 20 degC (68 degF).	 293.15	33.3 293.15	% K

Element :	Percent Min:	Percent Max:
Copper (Cu)	99.99	99.99

MaterialType :

Metal

Metal, Non-Ferrous

Alloy

FIGURE 2.25 PKS-MPD Material Properties Print Preview Page 2 for Copper (UNS C10100)

FIGURE 2.26 PKS-MPD Materials selection Properties display page for Copper (UNS C10100)

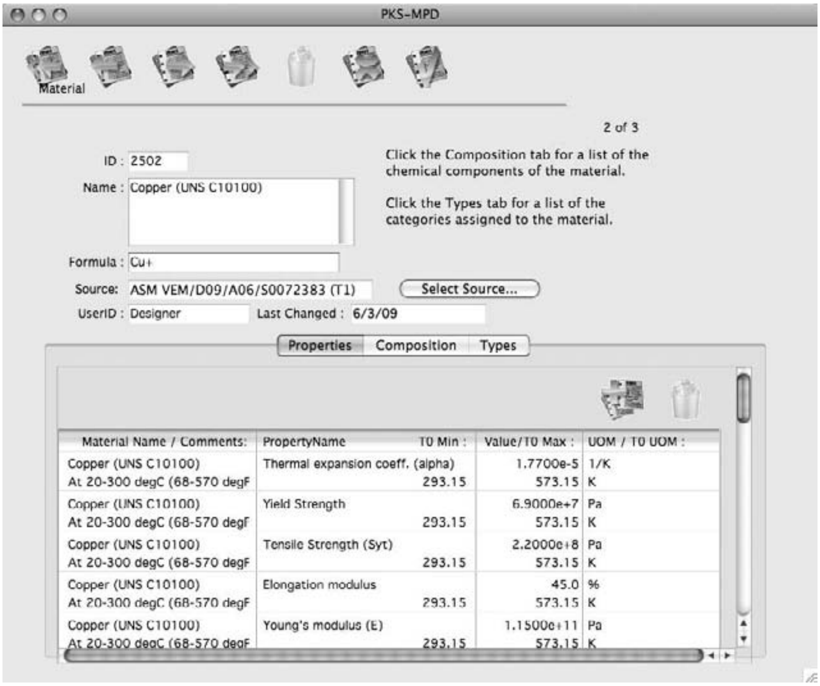


FIGURE 2.27 PKS-MPD Materials selection Properties display page for Copper (UNS C10100)

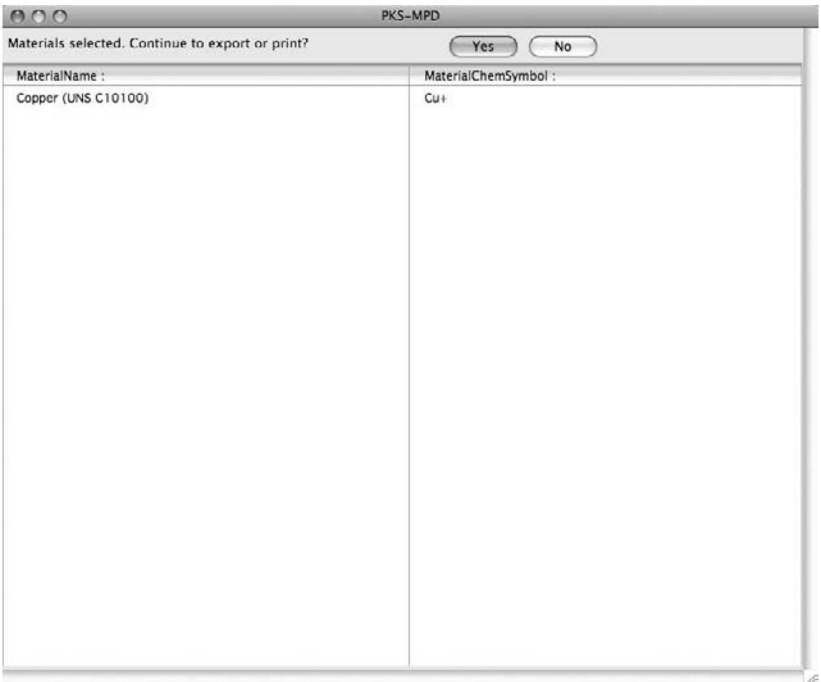


FIGURE 2.28 PKS-MPD Materials selected verification page for Copper (UNS C10100)

FIGURE 2.29 PKS-MPD Materials selection Tensile Strength (Syt) display page for Copper (UNS C10100)

16. Select the Tensile Strength (Syt) for Copper (UNS C10100) at room temperature.
17. Click Exit.

Continue the selection process for each property as displayed. When finished, click the Exit button for each property. The file is then exported as a text file with library management data leading. Figure 2.30 shows the material properties exported.

The exported file can be directly imported into COMSOL Multiphysics as follows:

1. Open COMSOL Multiphysics in the application mode of choice.
2. Using the menu bar, select Options > Materials/Coefficients Library.
3. Click the Add Library button. See Figure 2.31.
4. Select the newly exported Copper (UNS C10110) library. See Figure 2.32.

FIGURE 2.30 PKS-MPD Materials selection properties for Copper (UNS C10100) exported

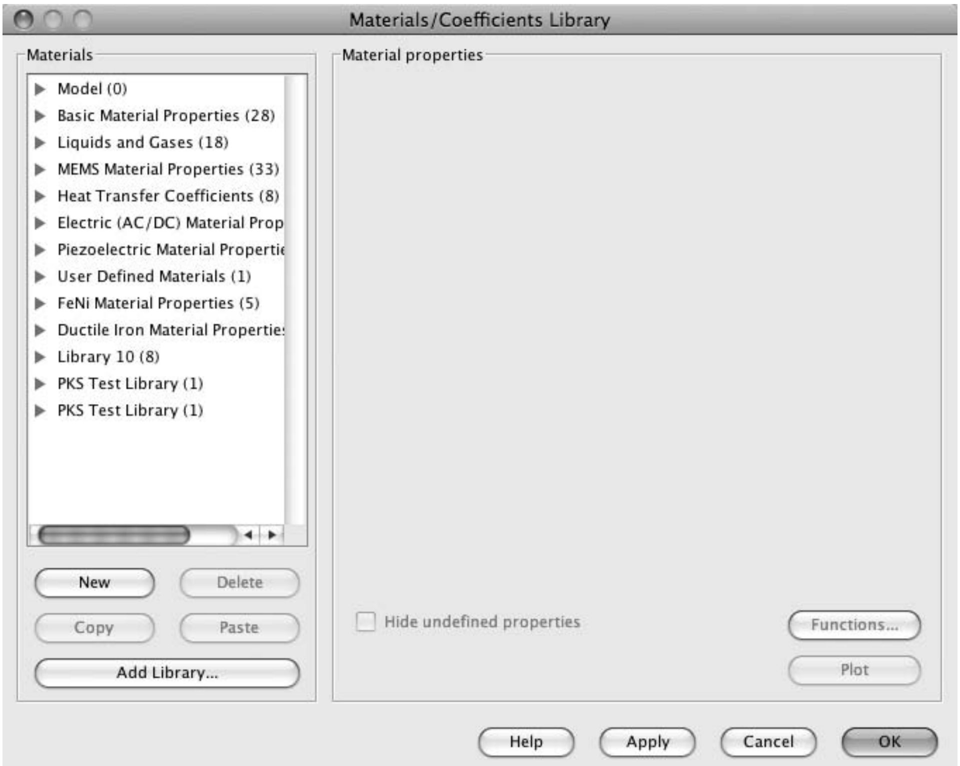


FIGURE 2.31 Materials/Coefficients Library edit page

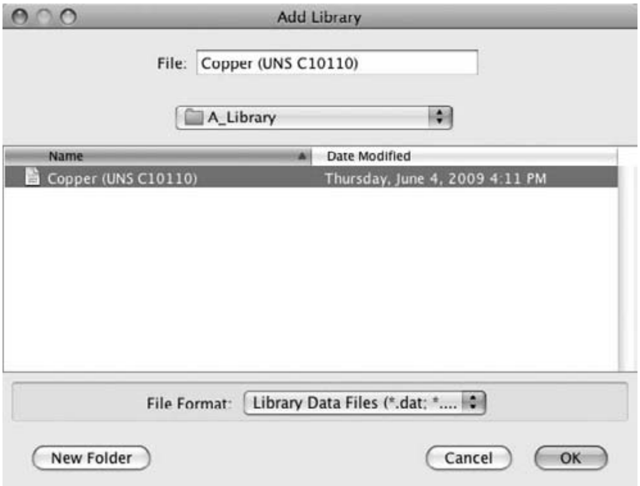


FIGURE 2.32 Copper (UNS C10100) file selected as the library to be added

5. Click OK. The Copper (UNS C10100) (1) library is added as the last item on the Materials library list in the Materials window. See Figure 2.33.
6. Click OK.

The second method of Searching if the modeler knows the UNS number, is simply to enter that number.

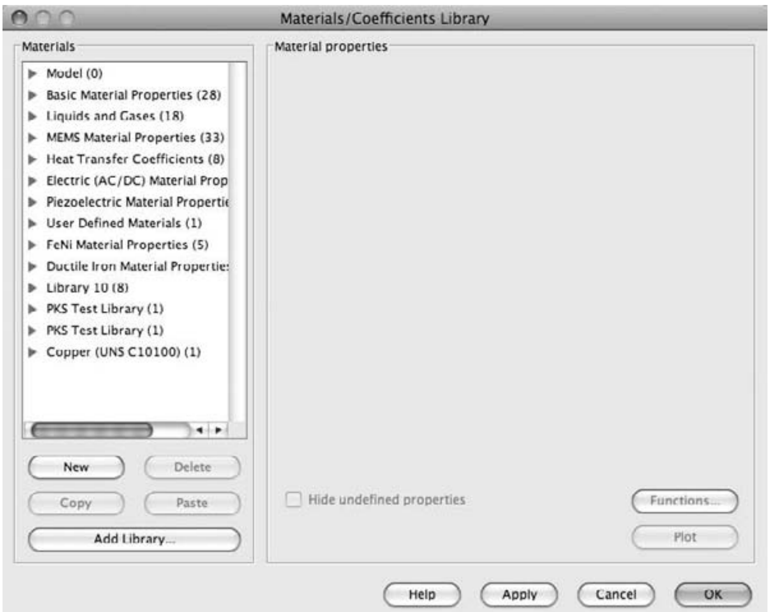


FIGURE 2.33 Materials/Coefficients Library edit page

FIGURE 2.34 PKS-MPD Materials selected Other Characteristics page

1. Click the Clear button.
2. Select “Other Characteristics.”
3. Enter @UNS C10100@.
4. Click the Add to Search button. See Figure 2.34.
5. The search yields one candidate material. Click the Print Materials button. See Figure 2.35.

The rest of the instructions for printing, exporting, and adding materials properties to the COMSOL library are the same as given previously.

FIGURE 2.35 PKS-MPD Materials selected Properties page

References

1. http://www.comsol.com/products/material/
2. http://en.wikipedia.org/wiki/DIN
3. http://en.wikipedia.org/wiki/Unified_numbering_system
4. http://en.wikipedia.org/wiki/Oxygen-free_copper
5. http://www.matweb.com/
6. http://www.pks-mpd.com/
7. Contact Pryor Knowledge Systems, Inc., at http://www.pks-mpd.com for a PKS-MPD sample database and an activation key.

Exercises

1. Explore the processes of finding and exporting materials properties with the COMSOL Material Library module presented in this chapter.
2. Explore the processes of finding and exporting materials properties with MatWeb as presented in this chapter.
3. Explore the processes of finding and exporting materials properties with PKS-MPD as presented in this chapter.

3 1D Modeling

In This Chapter

■ 1D Guidelines for New COMSOL® Multiphysics® Modelers

1D Modeling Considerations

1D modeling is both the least difficult and potentially the most difficult type of model to build, irrespective of the modeling software utilized. The least difficult aspect of 1D model building arises from the fact that the geometry is simple: In a 1D model, the modeler can have only a single line or a sequence of line segments as the modeling space. However, the physics in a 1D model can range from reasonably easy (simple) to extremely difficult (complex).

NOTE COMSOL® Multiphysics® software has two 1D modes: 1D (beginning-level through moderate-level modeling) and 1D Axisymmetric (advanced-level modeling). In keeping with the introductory focus of the material in this text, only 1D models (beginning-level through moderate-level models) will be presented. For information on the 1D Axisymmetric geometry, the associated physics, and the use of the same,

refer to the COMSOL manuals, the COMSOL website, and the general COMSOL Multiphysics software-related research literature.

The 1D model implicitly assumes that the energy flow, the materials properties, the environment, and any other conditions and variables that are of interest are homogeneous, isotropic, and constant, unless otherwise specified, throughout the entire domain of interest, both within the model and in the environs of the model. In other words, the properties assigned to the 1D model are representative of the properties of proximate nonmodeled regions. Bearing that in mind, the modeler needs to ensure that all modeling conditions and associated parameters have been properly considered, defined, or set to the appropriate values.

NOTE As mentioned earlier, it is always preferable for the modeler to be able to accurately anticipate the expected behavior (results) of the model. Calculated solution values that deviate widely from the expected values or from comparison values measured in experimentally derived realistic models are probably indicative of one or more modeling errors either in the original model design, in the earlier model analysis, or in the understanding of the underlying physics, or are simply due to human error.

Coordinate System

In a 1D model, there are only two coordinates: space (x) and time (t). In a steady-state solution, parameters vary only as a function of space (x). In a transient solution model, parameters can vary both in space (x) and in time (t). The space coordinate (x) typically represents distance throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current).

NOTE To assist the reader to achieve a broader exposure to the applicability of physics discussed in this book and to demonstrate the power of 1D modeling techniques, modeling examples are presented that demonstrate techniques from two different, but similar, broadly applicable areas of physics. The examples presented explore wave propagation, in the broadest general sense.

■ 1D KdV Equation: Solitons and Optical Fibers

The KdV equation[1] is a well-known example of a group of nonlinear partial differential equations[2] called exactly solvable.[3] That type of equation has solutions that can be specified with exactness and precision.

NOTE Nonlinear partial differential equations play an extremely important role in the description of physical systems.[4] Nonlinear partial differential equations are, by and large, inherently difficult to solve and require a unique approach for each equation type.

The KdV equation, solved in 1895 by Diederik Korteweg and Gustav de Vries, mathematically describes the propagation of a surface disturbance on a shallow canal. The effort to solve this wave propagation problem was undertaken based on observations by John Scott Russell in 1834,[5] among others. Subsequent activity in this mathematical area has led to soliton application in magnetics[6] and optics.[7] Work on soliton propagation problems is currently an active area of research.

The following numerical solution model (KdV equation) was originally developed by COMSOL for distribution with the Multiphysics software as an equation-based model. Here, we will build the model as presented in the COMSOL Model Library and then explore variations and expansions on the model.

NOTE It is important for the new modeler to personally try to build each model presented within the text. There is no substitute for the hands-on experience of actually building, meshing, solving, and postprocessing a model. Many times the inexperienced modeler will make and subsequently correct errors, adding to his or her experience and fund of modeling knowledge. Even the simplest model will expand the modeler's fund of knowledge.

The KdV equation (as written in standard notation) is

$$\partial_t u + \partial_x^3 u + 6u\partial_x u = 0 \tag{3.1}$$

In the COMSOL documentation, the formula is shown as

$$u_t + u_{xxx} = 6uu_x \quad \text{in} \quad \Omega = [-8, 8] \tag{3.2}$$

The difference between the two equations is that (3.2) is the negative form of (3.1), which will be adjusted during postprocessing.

The boundary conditions are periodic, as shown here:

$$u(-8, t) = u(8, t) \quad \text{periodic} \tag{3.3}$$

The initial condition for this model is

$$u(x, 0) = -6\text{sec}h^2(\text{x}) \tag{3.4}$$

Once the modeler builds and solves this model, it will be seen that the pulse immediately divides into two soliton pulses, with different width and propagation speeds.

COMSOL KdV Equation Model

To start building the KdV_Equation_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "1D" from the Space dimension pull-down list. Select COMSOL Multiphysics > PDE Modes > PDE, General

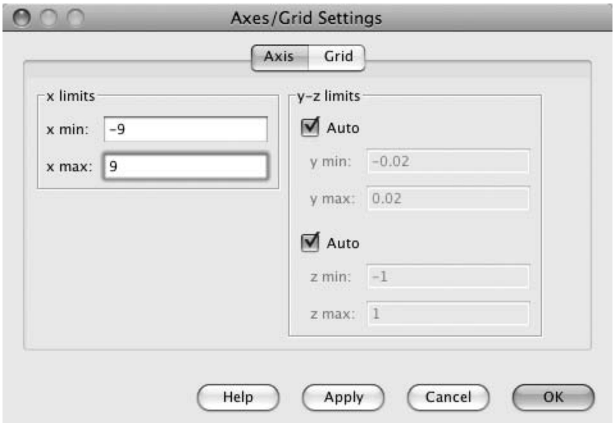


FIGURE 3.1 1D Axes/Grid Settings window (*x*)

Form > Time-dependent analysis. Type u1 space u2 in the dependent variables edit field. Click OK. Using the menu bar, select Options > Axes/Grid Settings. Enter –9 tab 9 in the edit fields to define the *x* geometry. Click OK. See Figure 3.1.

Using the menu bar, select Draw > Specify Objects > Line. Enter –8 space 8 in the Line edit window. Click OK. See Figure 3.2.

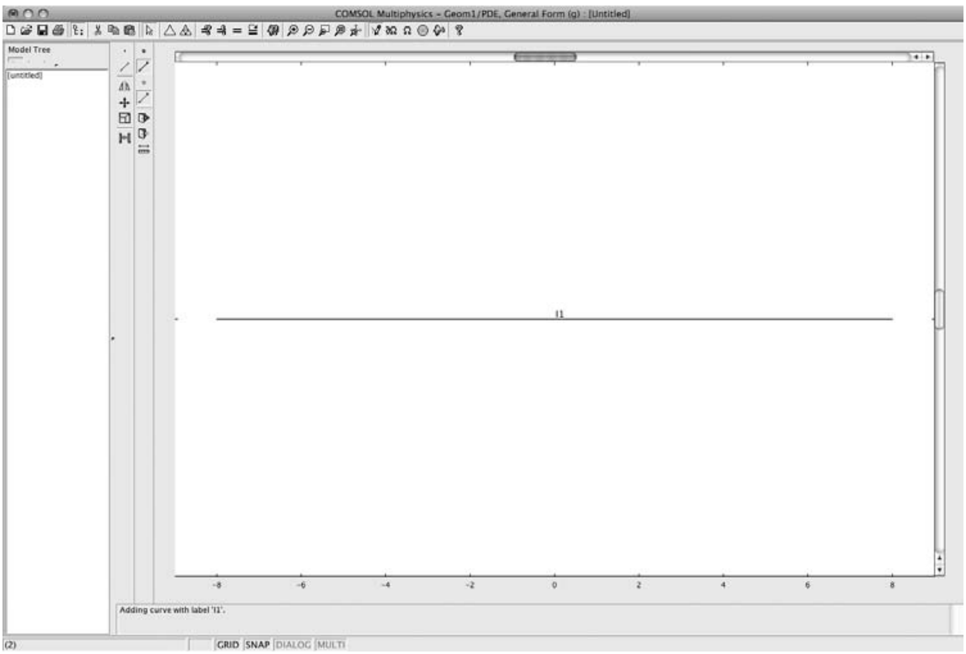


FIGURE 3.2 1D geometry for the KdV equation model

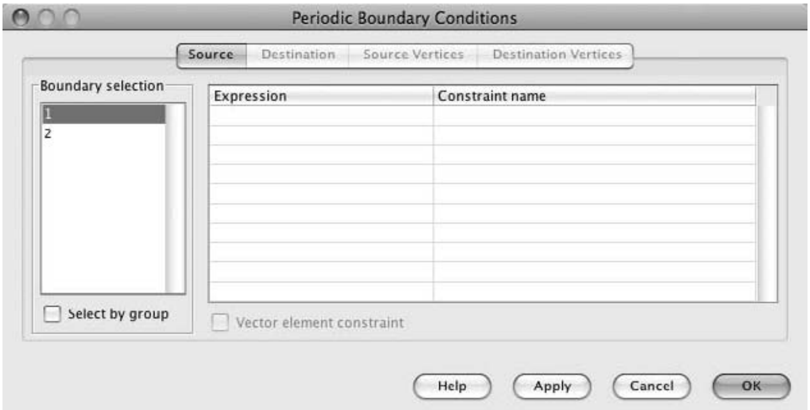


FIGURE 3.3 Periodic Boundary Conditions window

Periodic Boundary Condition Settings

NOTE For the new modeler unfamiliar with periodic boundary conditions, their use allows the domain (x values) of the model to be extended essentially indefinitely. For example, the modeling workspace of a line has two ends that would form two abrupt terminations, if not somehow compensated for. The use of periodic boundary conditions forms the line into a circle, which is mathematically infinite (endless).

Having established the 1D geometry (line), the next step is to define the fundamental physics conditions. From the menu bar, select Physics > Periodic Conditions > Periodic Boundary Conditions. After the Periodic Boundary Conditions window appears, on the Source page, select "1" in the Boundary selection window. See Figure 3.3.

Enter u1 in the Expression edit window, and then press the Enter key. The constraint name "pconstr1" will appear in the Constraint name column. See Figure 3.4.

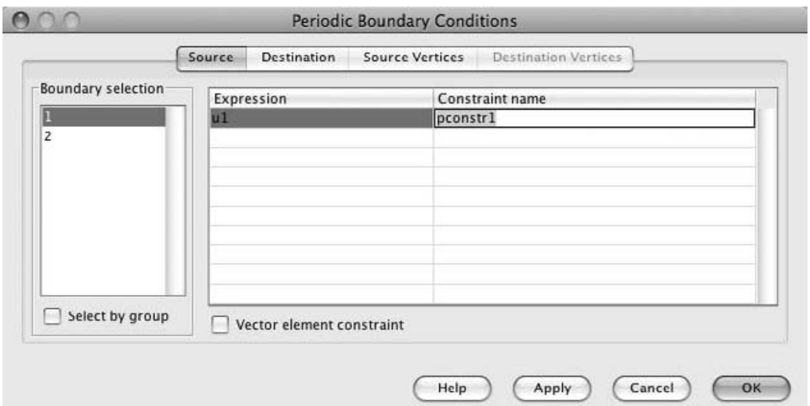


FIGURE 3.4 Periodic Boundary Conditions window, Source page

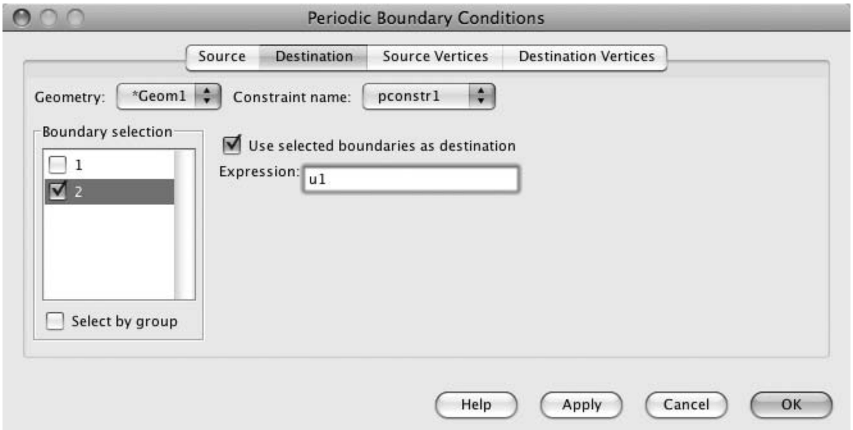


FIGURE 3.5 Periodic Boundary Conditions window, Destination page, boundary 2

Click the Destination tab. Select “2” as the boundary, and enter u1 in the edit window. See Figure 3.5.

Click the Source Vertices tab. Select “1” as the vertex, and then click the >> button. See Figure 3.6.

Click the Destination Vertices tab. Select “2” as the vertex, and then click the >> button. See Figure 3.7.

Click the Source tab. Select “1” as the boundary, and then type u2 in the second Expression window. Press the Enter key. The label “pconstr2” will appear in the Constraint name column. See Figure 3.8.

Click the Destination tab. Select “2” as the boundary, and then enter u2 in the Expression edit window. See Figure 3.9.

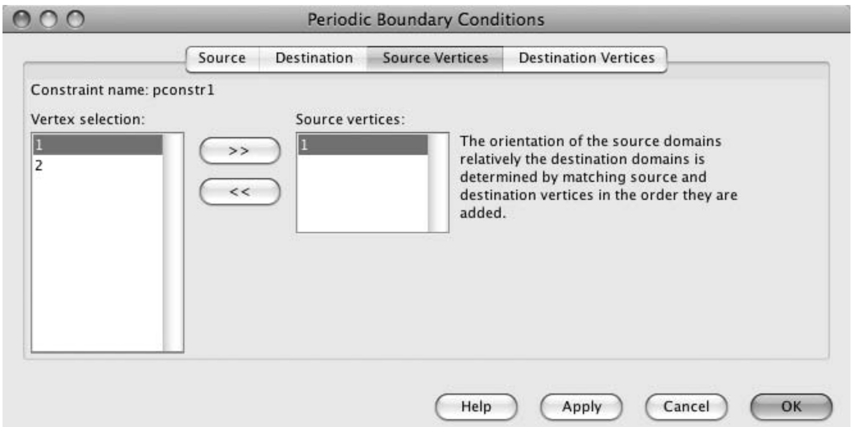


FIGURE 3.6 Periodic Boundary Conditions window, Source Vertices page, vertex 1

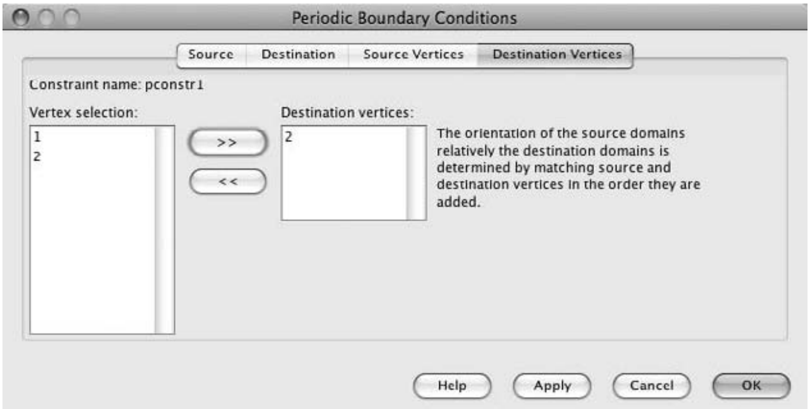


FIGURE 3.7 Periodic Boundary Conditions window, Destination Vertices page, vertex 2

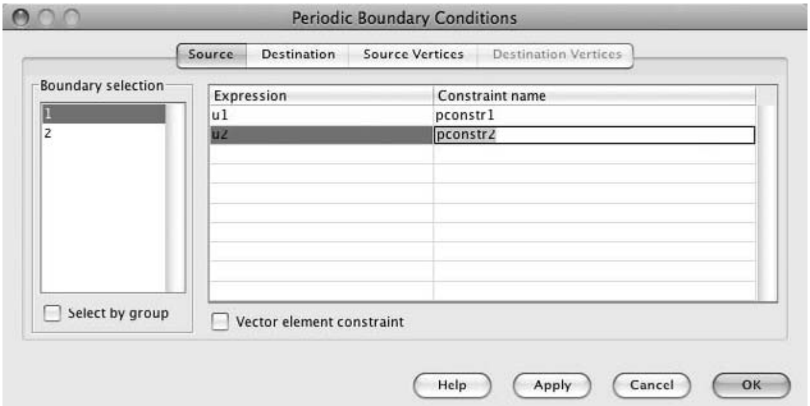


FIGURE 3.8 Periodic Boundary Conditions window, Source page, boundary 1, variable u2

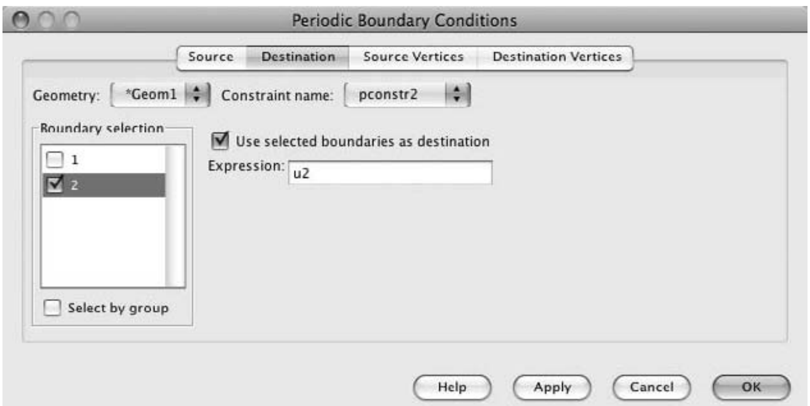


FIGURE 3.9 Periodic Boundary Conditions window, Destination page

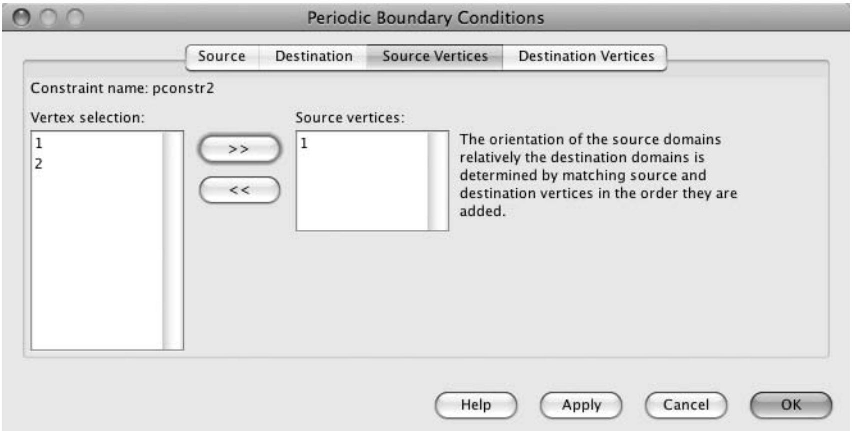


FIGURE 3.10 Periodic Boundary Conditions window, Source Vertices page

Click the Source Vertices tab. Select "1" as the vertex, and then click the >> button. See Figure 3.10.

Click the Destination Vertices tab. Select "2" as the vertex, and then click the >> button. See Figure 3.11. Click OK.

Boundary Conditions Settings

The next step is to set the boundary conditions. Using the menu bar, select Physics > Boundary Settings. Using Table 3.1 as a guide, on the Type page, select boundaries 1 and 2. Click the Neumann button, and then click the G tab. Verify or type 0 in each edit window. Click OK.

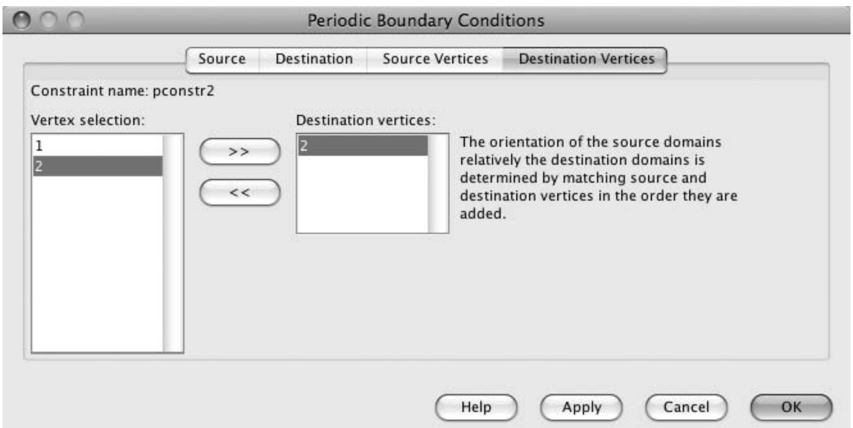


FIGURE 3.11 Periodic Boundary Conditions window, Destination Vertices page

Table 3.1 Boundary Settings Window

Parameter	Boundary 1	Boundary 2	G(1)	G(2)
Type	Neumann	Neumann		
Setting			0	0

NOTE Boundary conditions settings specify the values that a solution to the problem being solved needs to take on at the boundary (edge). Two types of boundary conditions are used in this book: Dirichlet and Neumann. In the Dirichlet boundary condition, $f(a) = n_1$ and $f(b) = n_2$, where a, b are the boundary points and n_1, n_2 are given numbers. In the Neumann boundary condition, $df(a)/dx = n_3$ and $df(b)/dx = n_4$, where a, b are the boundary points and n_3, n_4 are given numbers. Mixed boundary conditions, which are a more advanced topic and will not be covered here, are also possible.

Subdomain Settings

The next step in building the KdV equation model is to set the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select subdomain 1, and enter the coefficient values under the correct tab as shown in Table 3.2.

Once the PDE coefficients have been entered, Click the Init tab. Enter the initial conditions shown in Table 3.3, and then click OK.

Table 3.2 Subdomain Settings window, PDE coefficients

PDE Coefficient	**Value**
$\Gamma(1)$	u2
$\Gamma(2)$	u1x
$F(1)$	6*u1*u1x
$F(2)$	u2
$d_a(11)$	1
$d_a(12)$	0
$d_a(21)$	0
$d_a(22)$	0

Table 3.3 Initial Conditions Window

Initial Condition	**Value**
u1(t_0)	–6*sech(x)^2
u2(t_0)	–24*sech(x)^2*tanh(x)^2+12*sech(x)^2*(1-tanh(x)^2)

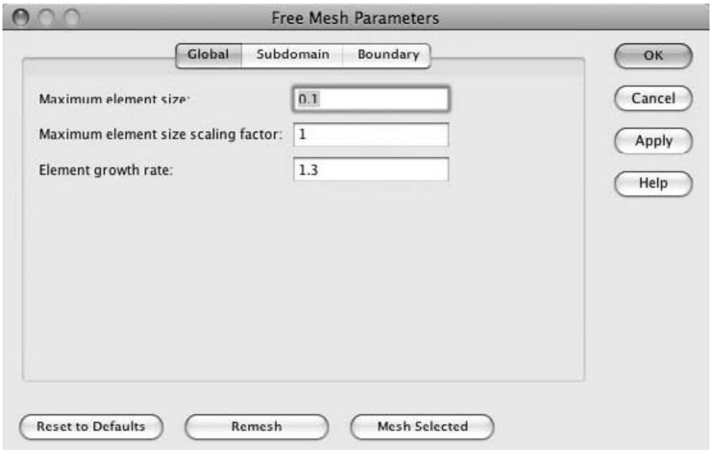


FIGURE 3.12 Free Mesh Parameters window

Mesh Generation

From the menu bar, select Mesh > Free Mesh Parameters. Type 0.1 in the Maximum element size edit window, as shown in Figure 3.12. The mesh consists of 160 elements. Click OK.

From the menu bar, select Mesh > Initialize Mesh. The result of the meshing operation is shown in Figure 3.13.

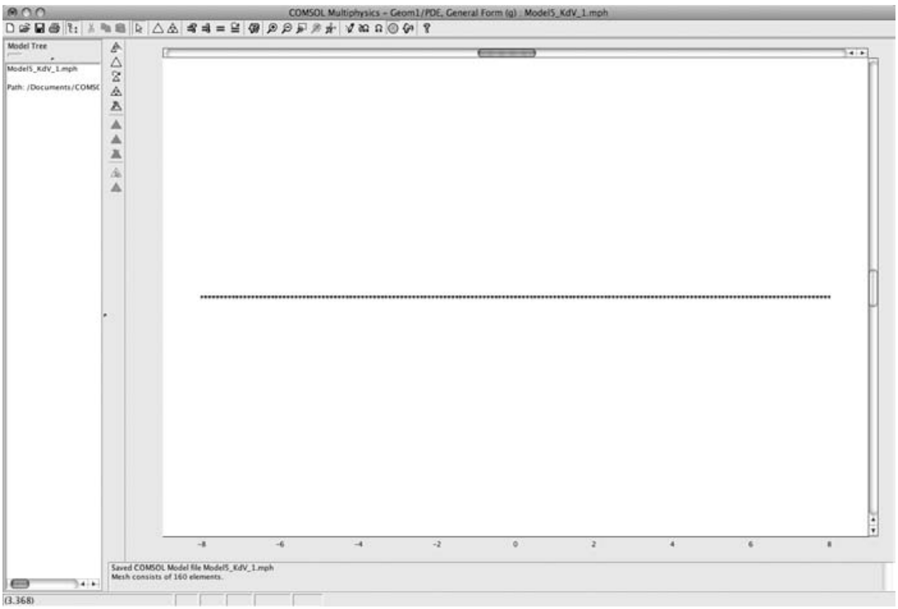


FIGURE 3.13 Meshed model

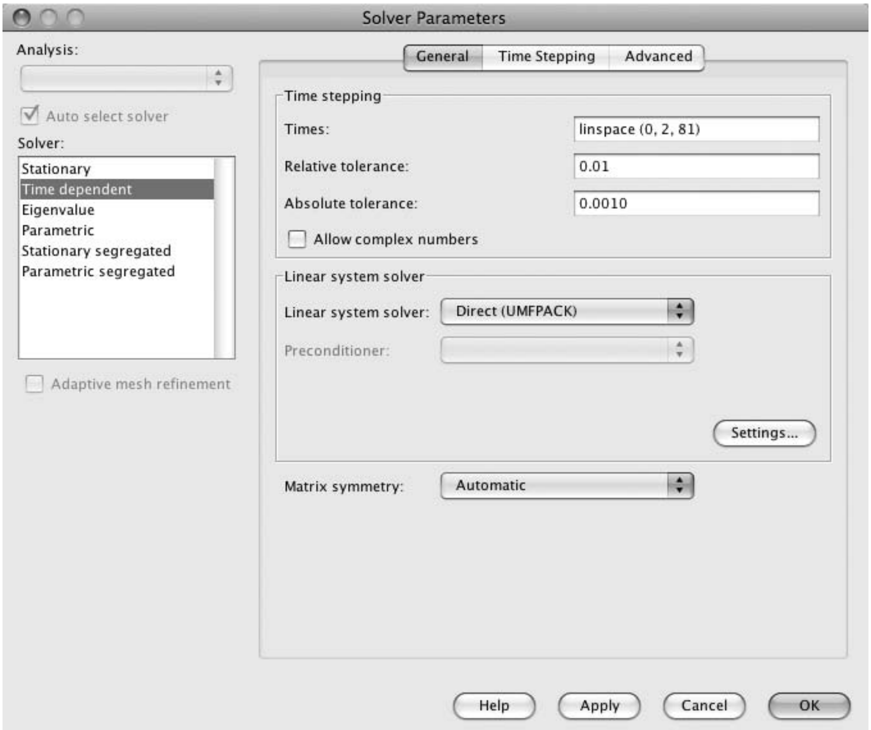


FIGURE 3.14 Solver Parameters window

Solving the KdV Equation Model

First, using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver. Type linspace(0, 2, 81) in the Times edit window, as shown in Figure 3.14. This instruction divides the time-space into 81 equal intervals, in the period from 0 to 2 seconds.

NOTE When the instruction linspace(a, b, c) is typed, it must be typed with *no* space between the last "e" of "linspace" and the open parenthesis (of the argument specification (a, b, c). If it is not typed exactly this way, the COMSOL Multiphysics software will indicate an error!

Click the Time Stepping tab. Type 2 in the Maximum BDF order edit window, as shown in Figure 3.15. Click OK.

Using the menu bar, select Solve > Solve Problem. The solution that is immediately seen is the negated (–) solution at the last time interval (t = 2 seconds), as shown in Figure 3.16.

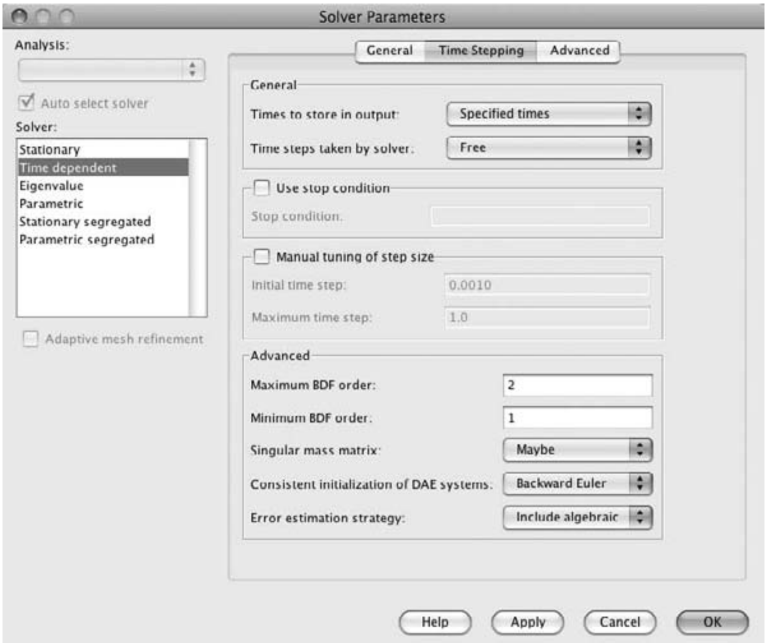


FIGURE 3.15 Solver Parameters window, Time Stepping page

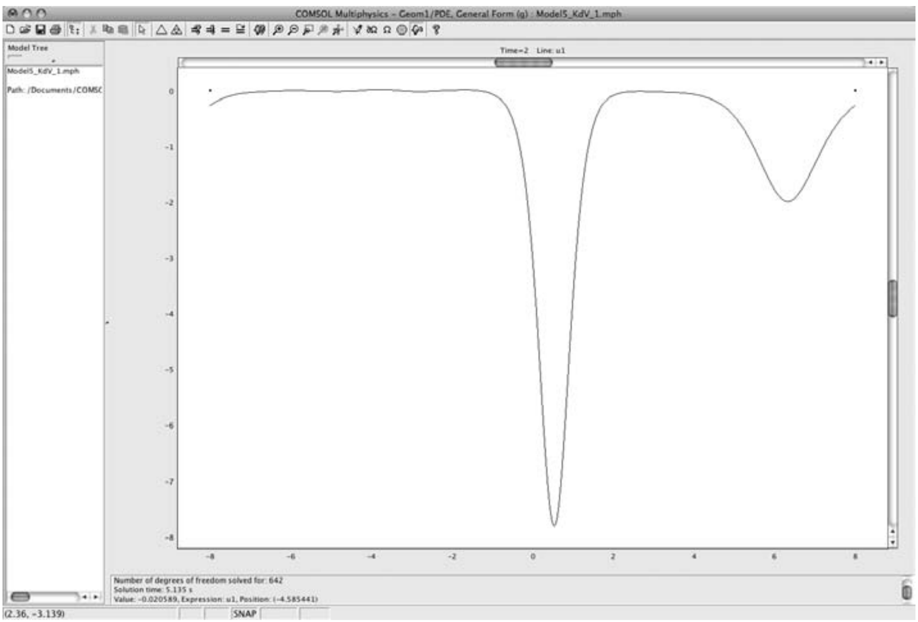


FIGURE 3.16 Negated KdV model solution

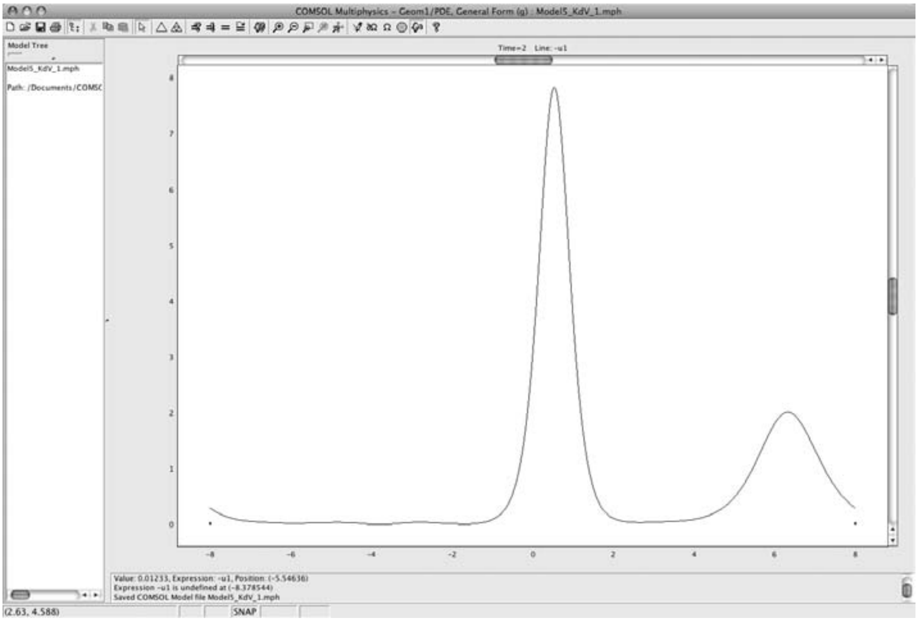


FIGURE 3.17 KdV equation model solution

Postprocessing

The positive solution can be viewed as follows: Select Postprocessing > Plot Parameters. Click the Line tab. Type –u1 in the Expression edit window. Click the Apply button. The positive results are shown in Figure 3.17.

Save the KdV Equation model as Model3_KdV_1.

The solution to the KdV equation can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears (see Figure 3.18), click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 3.19). Click the Start Animation button. Save the KdV equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3_KdV_1.avi that was supplied with this book.

NOTE Many modelers are better able to understand the dynamics of the solution when the solution is presented as an animation. It is available in addition to the presentation of the solution as a series of static plots.

The file extension that is created during the Save operation is platform dependent. If the platform is a Power Mac® computer, the extension for an animation will be different (.mov) than that for a Mac® computer with Intel® processor or a PC (.avi). Either can be played using a free QuickTime® player (http://www.apple.com/quicktime).

FIGURE 3.18 KdV model solution Plot Parameters window, Line page

FIGURE 3.19 KdV model solution, Animate page

First Variation on the KdV Equation Model

The previous solution to the KdV equation results in two soliton pulses propagating in the same medium at the same time. Next, we will explore how the model behaves when the initial conditions are modified. In this case, the argument is made smaller.

NOTE Information transmission relies on the measurement of a difference. In Morse code (a time differentiation method), the difference is between a long pulse, a short pulse, and no pulse. No pulse signifies no message. Thus, even if a message was sent, if it was not received (detected), then the recipient of the non-message classifies the message traffic as zero. To receive a message, the received signal must be of adequate amplitude (analog), of adequate duration (time), and in the expected frequency band of the receiver. The signal amplitude must be sufficiently greater than the detection threshold to allow information to be collected. The signal-to-noise ratio determines the minimum detectable signal.[8]

The stable, long-distance, light pulses used to convey information through optical fibers are known as temporal solitons.[9] To achieve detectability, the fiber is designed to compensate for dispersion (frequency spreading) and power loss.

First, save a copy of the just-created KdV equation model as Model3_KdV_2. You can then modify the KdV equation model without being concerned about damaging the original model.

If Model3_KdV_2 is already open on your desktop, skip to the "Scalar Expressions" section. If Model3_KdV_2 is not already open on your desktop, using the menu bar, select File > Open. When the Open Model window is displayed as in Figure 3.20, select "Model3_KdV_2." Click OK.

Scalar Expressions

Using the menu bar, select Options > Expressions > Scalar Expressions. When the Scalar Expressions window opens, type x_a in the Name column and x/1.33 in the Expression column, as shown in Figure 3.21. Click OK.

NOTE The scalar expression that was just created will be used as the new argument for the initial conditions of the KdV equation model.

Having created the new scalar variable x_a, the next step is to modify the Initial Conditions expression(s).

Changing the Subdomain Settings

The next step in modifying the KdV equation model is to change the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window

FIGURE 3.20 Open Model window

appears, select subdomain 1. Verify that the coefficient value under each of the indicated tabs is as shown in Table 3.4.

Once the PDE coefficients have been verified, click the Init tab. Either modify the existing equations or enter the initial conditions shown in Table 3.5, and then click OK.

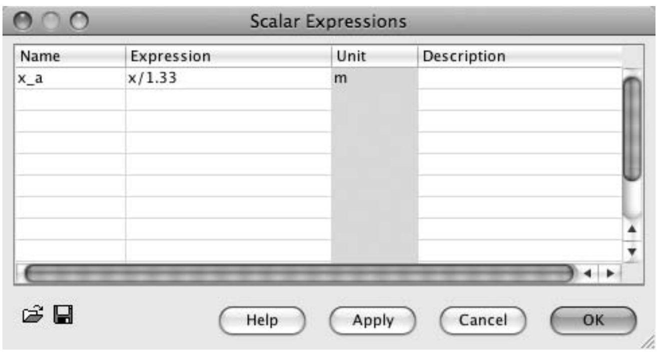


FIGURE 3.21 Scalar Expressions window

Table 3.4 Subdomain Settings Window, PDE Coefficients

PDE Coefficient	Value
$\Gamma(1)$	u2
$\Gamma(2)$	u1x
$F(1)$	6*u1*u1x
$F(2)$	u2
$d_a(11)$	1
$d_a(12)$	0
$d_a(21)$	0
$d_a(22)$	0

Because the new KdV equation model is a revised copy of the original KdV equation model, the new model will need to be reset. Using the menu bar, select File > Reset Model.

NOTE The Reset Model command clears the copied model of previous meshes and solutions.

Mesh Generation

From the menu bar, select Mesh > Free Mesh Parameters. Verify or type 0.1 in the Maximum element size edit window, as shown in Figure 3.22. Click OK.

From the menu bar, select Mesh > Initialize Mesh. The result of the meshing operation is shown in Figure 3.23.

Solving the First Revised KdV Equation Model

Using the menu bar, select Solve > Solver Parameters. Verify that the COMSOL Multiphysics software automatically selected the Time dependent solver. Verify or type linspace(0, 2, 81) in the Times edit window, as shown in Figure 3.24. This instruction divides the time-space into 81 equal intervals, in the period from 0 to 2 seconds.

Table 3.5 Initial Conditions Window

Initial Condition	Value
$u1(t_0)$	−6*sech(x_a)^2
$u2(t_0)$	−24*sech(x_a)^2*tanh(x_a)^2+12*sech(x_a)^2*(1-tanh(x_a)^2)

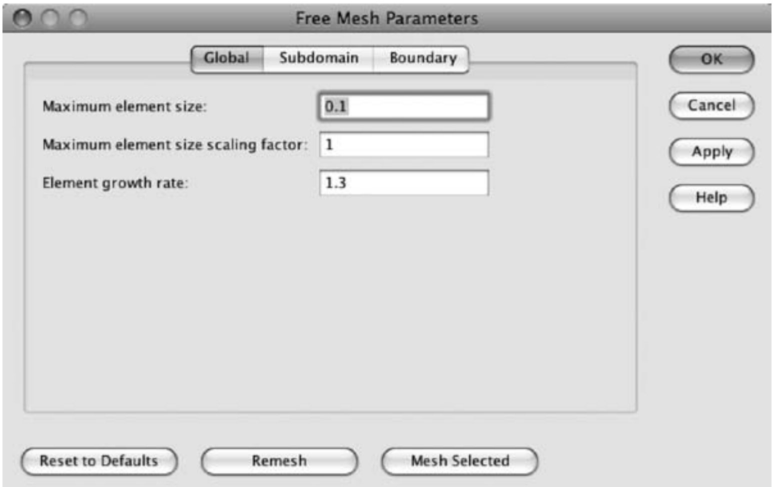


FIGURE 3.22 Free Mesh Parameters window

Click the Time Stepping tab. Verify or type 2 in the Maximum BDF order edit window, as shown in Figure 3.25. Click OK.

Using the menu bar, select Solve > Solve Problem.

In this variation, the solution that is immediately seen is not the negated (–) solution at the last time interval. Instead, the solution shown is the positive solution

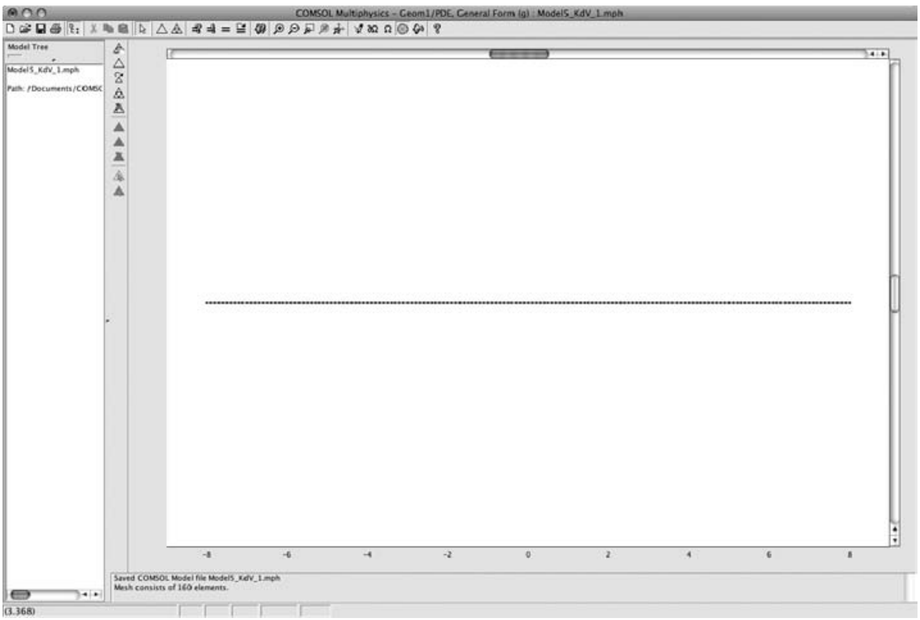


FIGURE 3.23 Remeshed model

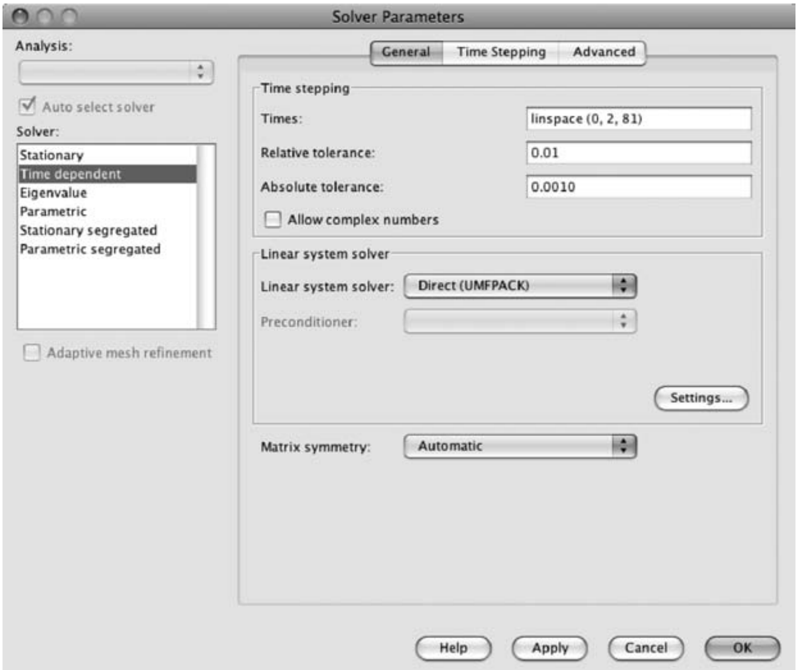


FIGURE 3.24 Solver Parameters window

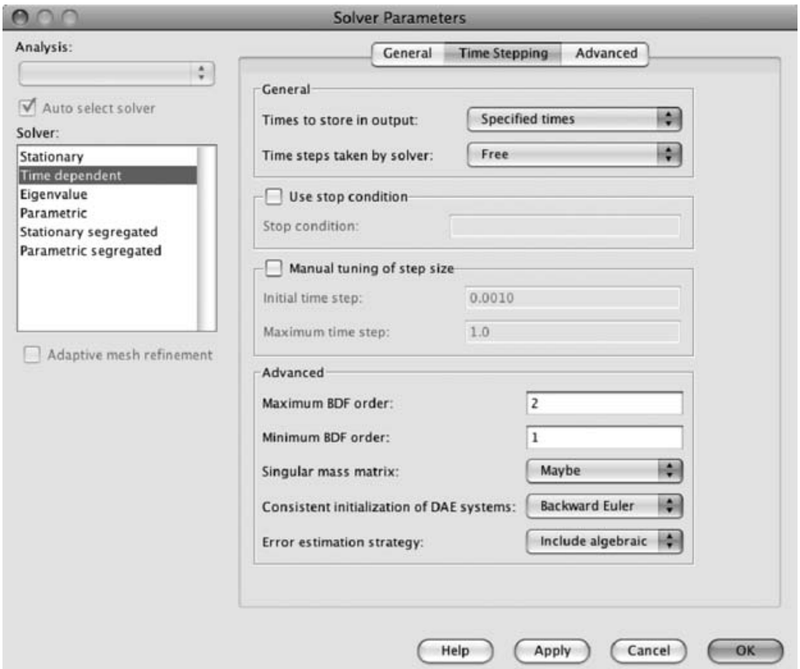


FIGURE 3.25 Solver Parameters window, Time Stepping page

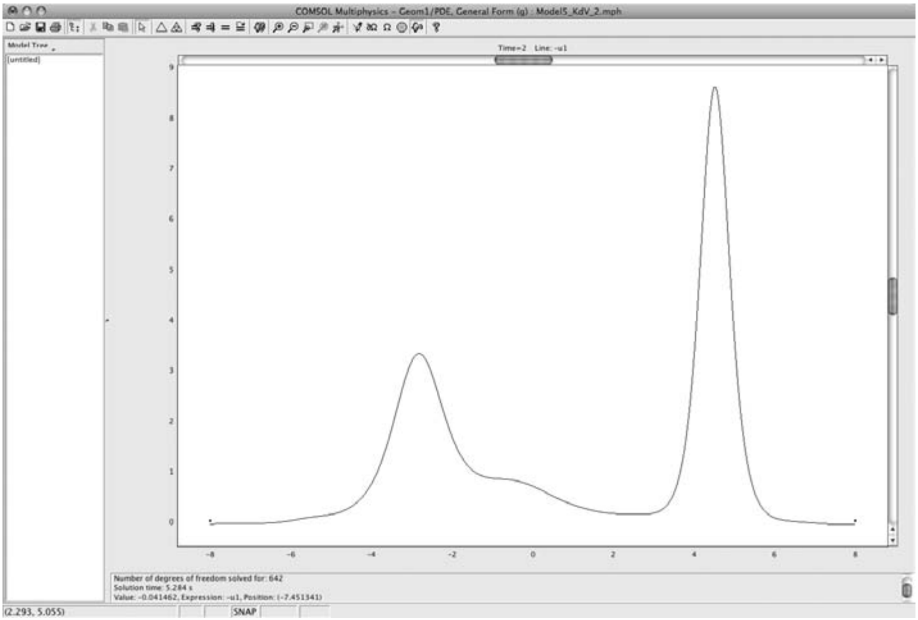


FIGURE 3.26 First revised KdV model solution

of the KdV Equation, because the sign inversion (–) was adjusted in postprocessing of the copied model. The results of the changed argument solution are shown in Figure 3.26.

Save this KdV Equation model as Model3_KdV_2 to retain the current solution.

Postprocessing Animation

This solution to the KdV equation can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 3.27). Click the Start Animation button. Save this KdV equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3_KdV_2.avi that was supplied with this book.

NOTE The reduction of the argument for the initial conditions results in the splitting of the initial, single soliton pulse into three separate soliton pulses that propagate through the medium (e.g., optical fiber) at three different velocities and arrive at the receiver at different times.

Adoption of either the first solution or the second solution in an information transmission system would cause serious message distortion or interference problems

FIGURE 3.27 KdV model solution, Animate page

at the receiving site. These solutions would cause the same nature of interference as multiple-path propagation in atmospheric transmission (e.g., the same signal arriving several times at the same receiver in a slightly delayed mode).

Second Variation on the KdV Equation Model

The first revised solution to the KdV equation results in three soliton pulses propagating in the same medium at the same time. Next, we will explore how the model behaves when the initial conditions are again modified. In this case, the argument of the initial conditions will be increased in size.

NOTE Remember—information transmission relies on the measurement of a difference. Each pulse is one bit of information. No pulse signifies no message. Thus, even if a message was sent, if it was not received (detected), then the recipient of the non-message classifies the message traffic as zero.

To receive the correct message, the signal must be of adequate amplitude (analog), of adequate duration (time), in the expected frequency band of the receiver, and must correlate exactly with the message sent. The signal amplitude must be sufficiently greater than the detection threshold to allow information to be collected and must not contain spurious, random pulses.

FIGURE 3.28 Open Model window

First, save a copy of the just-created first variation on the KdV equation model as Model3_KdV_3. You can then modify the KdV equation model without being concerned about damaging the just-built model.

If Model3_KdV_3 is already open on your desktop, skip to the "Scalar Expressions" section. If Model3_KdV_3 is not already open on your desktop, using the menu bar, select File > Open. When the Open Model window is displayed as in Figure 3.28, select "Model3_KdV_3." Click OK.

Scalar Expressions

Using the menu bar, select Options > Expressions > Scalar Expressions. When the Scalar Expressions window opens, verify or type x_a in the Name column and pi/2*x in the Expression column, as shown in Figure 3.29. Click OK.

NOTE The scalar expression that was just created will be used as the new larger argument for the initial conditions of the second variation on the KdV equation model.

Having created the new Scalar Variable x_a, the next step is to modify the Initial Conditions expression(s).

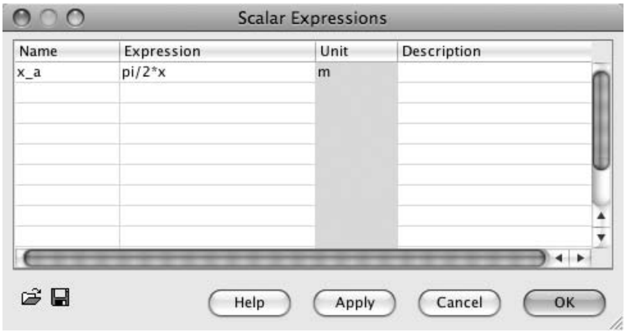


FIGURE 3.29 KdV Scalar Expressions window

Changing the Subdomain Settings

The next step in modifying this version of the KdV equation model is to change the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select subdomain 1. Verify that the coefficient value assigned to each of the indicated tabs is as shown in Table 3.6.

Once the PDE coefficients have been verified, click the Init tab. Verify the existing equations or type the initial conditions found in Table 3.7, and then click OK.

Table 3.6 Subdomain Settings Window, PDE Coefficients

PDE Coefficient	Value
$\Gamma(1)$	u2
$\Gamma(2)$	u1x
$F(1)$	6*u1*u1x
$F(2)$	u2
$d_a(11)$	1
$d_a(12)$	0
$d_a(21)$	0
$d_a(22)$	0

Table 3.7 Initial Conditions Window

Initial Condition	Value
$u1(t_0)$	–6*sech(x_a)^2
$u2(t_0)$	–24*sech(x_a)^2*tanh(x_a)^2+12*sech(x_a)^2*(1-tanh(x_a)^2)

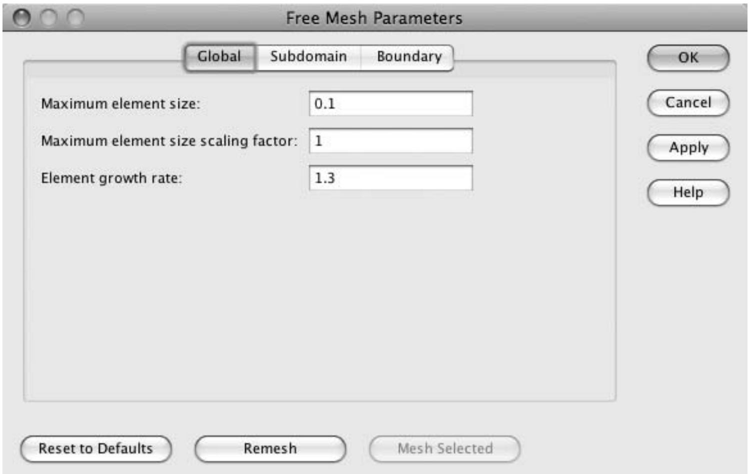


FIGURE 3.30 Free Mesh Parameters window

Because the new KdV equation model is a revised copy of the original KdV equation model, this model will need to be reset. Using the menu bar, select File > Reset Model.

NOTE The Reset Model command clears the copied model of previous meshes and solutions.

Mesh Generation

From the menu bar, select Mesh > Free Mesh Parameters. Verify or type 0.1 in the Maximum element size edit window, as shown in Figure 3.30. Click OK.

From the menu bar, select Mesh > Initialize Mesh. The result of the meshing operation is shown in Figure 3.31.

Solving the Second Revised KdV Equation Model

Using the menu bar, select Solve > Solver Parameters. Verify that the COMSOL Multiphysics software automatically selected the Time dependent solver. Verify or type linspace(0, 2, 81) in the Times edit window, as shown in Figure 3.32. This instruction divides the time-space into 81 equal intervals, in the period from 0 to 2 seconds.

Click the Time Stepping tab. Verify or type 2 in the Maximum BDF order edit window, as shown in Figure 3.33. Click OK.

Using the menu bar, select Solve > Solve Problem.

The solution that is immediately seen is not the negated (–) solution at the last time interval. Instead, the solution shown is the positive solution of the KdV equation,

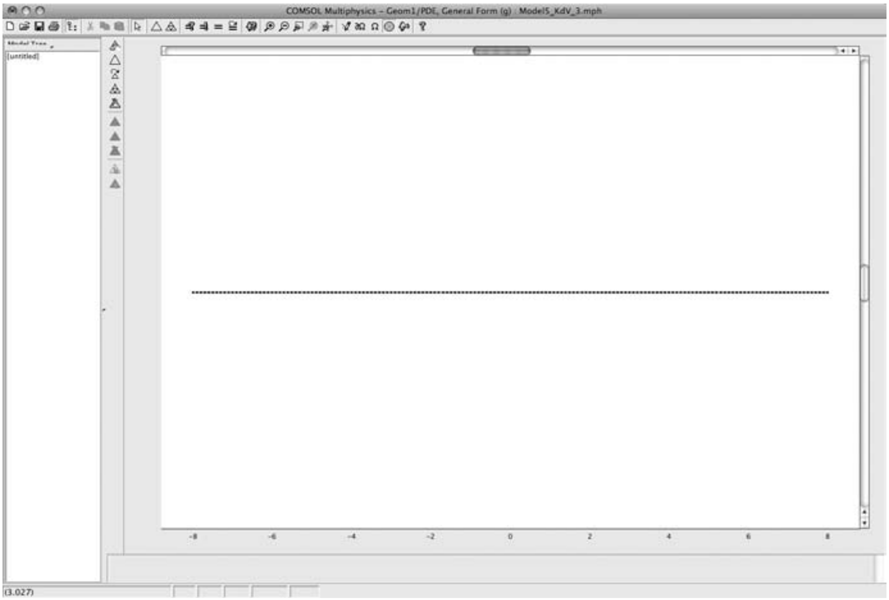


FIGURE 3.31 Remeshed model

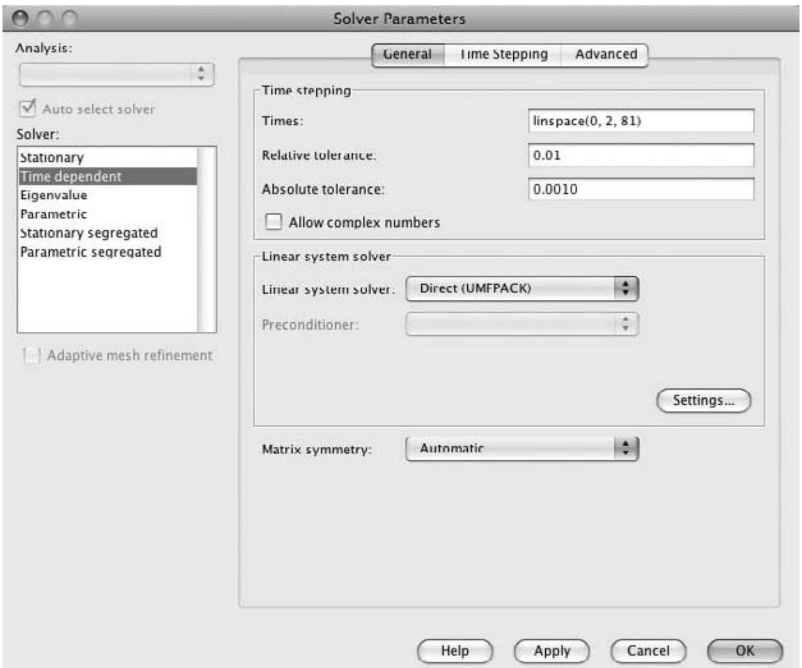


FIGURE 3.32 Solver Parameters window

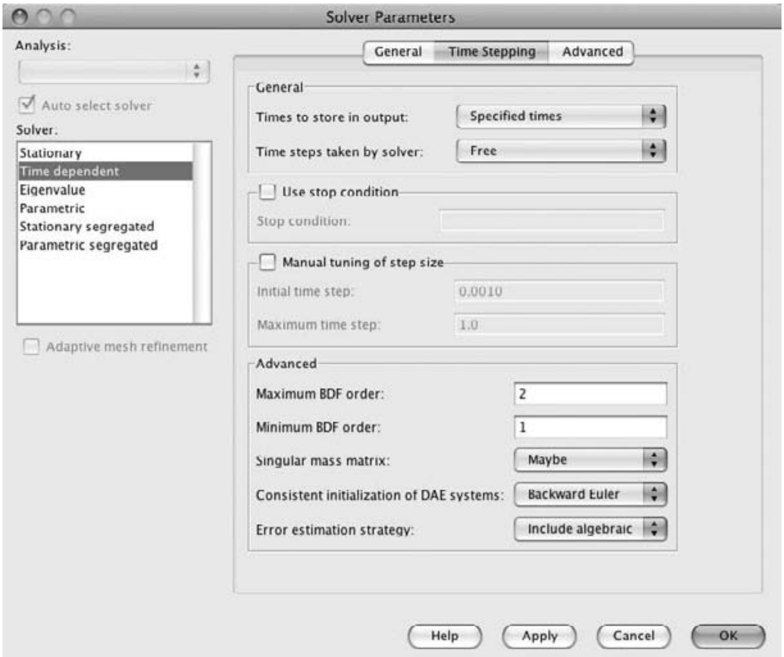


FIGURE 3.33 Solver Parameters window, Time Stepping page

because the sign inversion (–) was adjusted in postprocessing of the previous model. The results of the changed argument solution are as shown in Figure 3.34.

Save this KdV equation model as Model3_KdV_3 to retain the current solution.

Postprocessing Animation

This solution to the KdV equation can also be viewed as an animation. To view this solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 3.35). Click the Start Animation button. Save this KdV equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3_KdV_3.avi that was supplied with this book.

NOTE The result of the argument change for the second variation on the KdV equation model initial conditions is the generation of a single soliton that propagates through the medium (e.g., optical fiber) at one velocity. This soliton pulse will reliably convey information to the receiving station.

One factor that this model does not address is the loss of energy (attenuation) as a function of distance. It is a more advanced topic that will not be covered in this book.

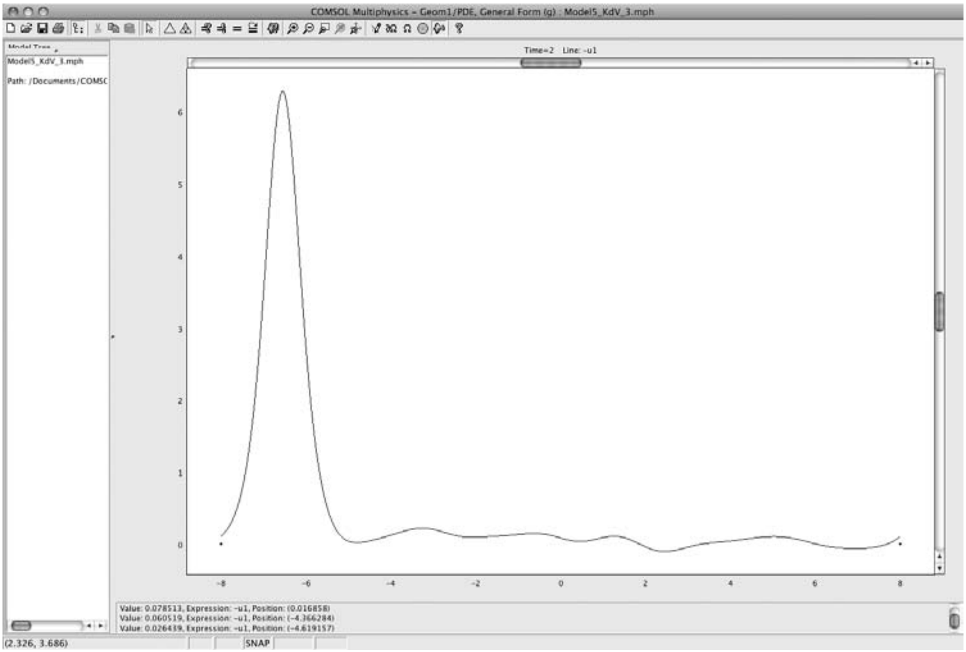


FIGURE 3.34 Second variation on the KdV equation model solution

FIGURE 3.35 KdV model solution, Animate page

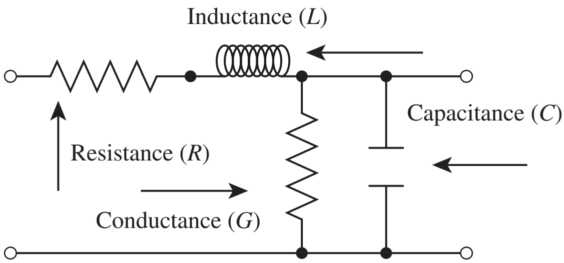


FIGURE 3.36 Telegraph equation electrical component model

1D KdV Equation Models: Summary and Conclusions

The KdV equation is a powerful tool that can be used to model soliton wave propagation in diverse media (e.g., physical waves in liquids, electromagnetic waves in transparent media). It is easily and simply modeled with a 1D PDE mode model.

1D Telegraph Equation

The telegraph equation[10] was developed by Oliver Heaviside[11] and first published about 1885.[12] The telegraph equation is based on a lumped constant, four-terminal electrical component model, as shown in Figure 3.36.

In this schematic model of the telegraph wires (and other transmission lines), there are four fundamental components: resistance (R) per unit of length (e.g., foot, meter), inductance (L) per unit of length (e.g., foot, meter), conductance (G) per unit of length (e.g., foot, meter), and capacitance (C) per unit of length (e.g., foot, meter). The differential equations for voltage (V) and current (I) have the same form, as shown in equations 3.5 and 3.6.

Equation 3.5 shows the partial differential equation for voltage (V):

$$\frac{\partial^2}{\partial x^2}V = LC\frac{\partial^2}{\partial t^2}V + (RC + GL)\frac{\partial}{\partial t}V + GRV \tag{3.5}$$

Equation 3.6 shows the partial differential equation for current (I):

$$\frac{\partial^2}{\partial x^2}I = LC\frac{\partial^2}{\partial t^2}I + (RC + GL)\frac{\partial}{\partial t}I + GRI \tag{3.6}$$

Equations 3.5 and 3.6 are similar in form to equation 3.7, as shown here for the COMSOL Multiphysics telegraph equation model:

$$u_{tt} + (\alpha + \beta)u_t + \alpha\beta u = c^2 u_{xx} \tag{3.7}$$

where α and β are positive constants, c is the transport velocity, and u is the voltage.

Equation 3.5 can be restated in subscript notation:

$$u_{xx} = LCu_{tt} + (RC + GL)u_t + GRu \tag{3.8}$$

Rearranging the terms of equation 3.7 gives the following equation:

$$u_{xx} = \frac{1}{c^2}u_{tt} + \frac{1}{c^2}(\alpha + \beta)u_t + \frac{1}{c^2}\alpha\beta u \tag{3.9}$$

Comparing equations 3.8 and 3.9 yields

$$LC = \frac{1}{c^2} \tag{3.10}$$

and

$$\alpha + \beta = \frac{(RC + GL)}{LC} \tag{3.11}$$

and

$$\alpha\beta = \frac{GR}{LC} \tag{3.12}$$

Solving for α and β:

$$\begin{aligned} \alpha &= \frac{CGL + C^2R - \sqrt{-4CGLR + (-CGL - C^2R)^2}}{2L} \\ \beta &= \frac{CGL + C^2R + \sqrt{-4CGLR + (-CGL - C^2R)^2}}{2L} \end{aligned} \tag{3.13}$$

or

$$\alpha = \frac{CGL + C^2R + \sqrt{-4CGLR + (-CGL - C^2R)^2}}{2L}$$

$$\beta = \frac{CGL + C^2R - \sqrt{-4CGLR + (-CGL - C^2R)^2}}{2L}$$

In the event that

$$R = G = 0 \tag{3.14}$$

the transmission line is considered lossless and the telegraph equation becomes

$$u_{xx} = LC\,u_{tt} \tag{3.15}$$

COMSOL 1D Telegraph Equation Model

Model Navigator

To start building the telegraph equation model, activate the COMSOL Multiphysics software. In the Model Navigator, select "1D" from the Space dimension pull-down list. Select COMSOL Multiphysics > PDE Modes > PDE, Coefficient Form > Time-dependent analysis, wave type. Verify that Lagrange-Quadratic elements have been selected in the Element pull-down list. Click OK.

1D Geometry

Once the COMSOL Multiphysics 1D workspace window has appeared, using the menu bar, select Draw > Specify Objects > Line. Type 0 space 1 in the Coordinates edit window of the Line window. Click OK. Using the toolbar, click the Zoom Extents button. The 1D geometry will appear as shown in Figure 3.37.

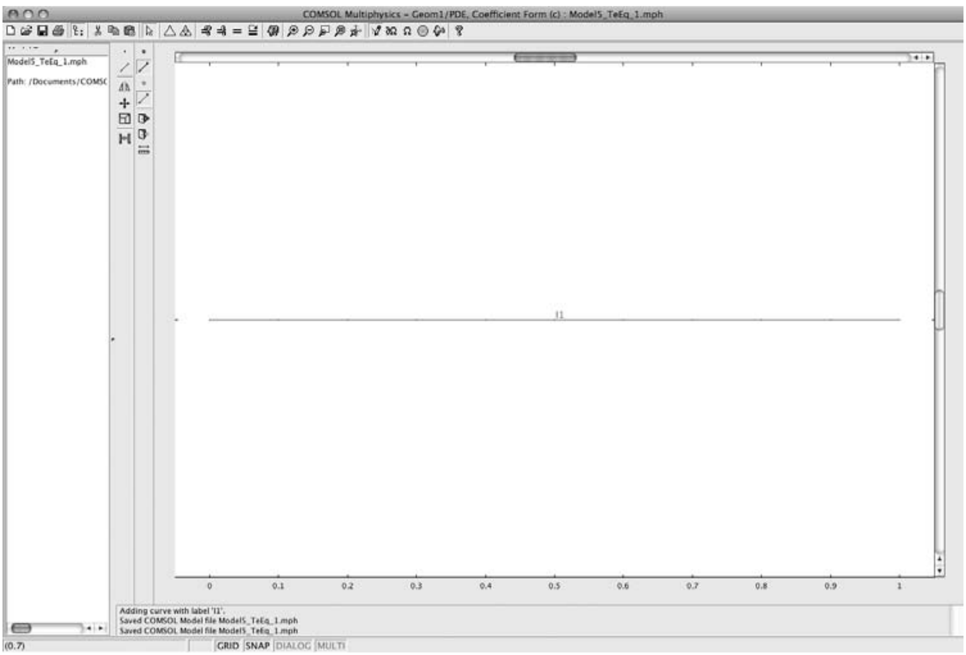


FIGURE 3.37 Telegraph equation geometry

Table 3.8 Constants Window

Name	Expression
c	1
alpha	0.25
beta	0.25

Table 3.9 Boundary Settings Window

Parameter	Boundary 1	Boundary 2	q	g
Type	Neumann	Neumann		
Setting			0	0

Constants

Using the menu bar, select Options > Constants. Type the constants in the Constants edit window, as indicated in Table 3.8, and then click OK.

Boundary Conditions

Using the menu bar, select Physics > Boundary Settings. After the Boundary Settings window appears, select both boundaries 1 and 2. Enter or verify the settings indicated in Table 3.9, and then click OK.

Subdomain Settings

The next step in building the telegraph equation model is to set the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select subdomain 1 and enter the coefficient values under the correct tab as shown in Table 3.10. Verify and then leave the other coefficient settings at their 0 value, as shown in Figure 3.38.

Table 3.10 Subdomain Settings Window, PDE Coefficients

PDE Coefficient	Value
c	c*c
a	alpha*beta
f	–(alpha+beta)*ut
e_a	1
d_a	0

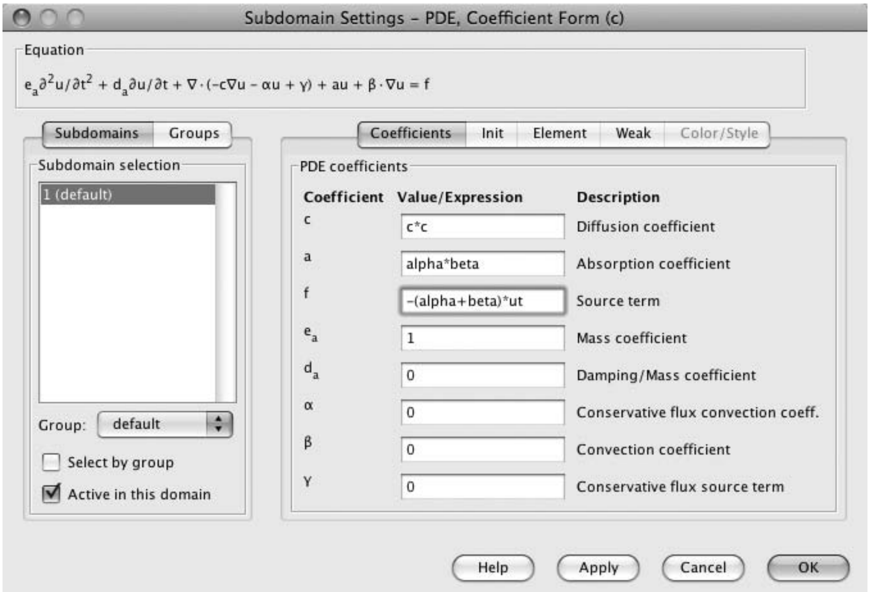


FIGURE 3.38 PDE window, Coefficients page

Once the PDE coefficients have been entered, click the Init tab. Enter the initial conditions found in Table 3.11, and then click OK. See Figure 3.39.

Mesh Generation

Using the toolbar, select Initialize Mesh > Refine Mesh once. The final mesh, with 30 elements, is shown in Figure 3.40.

Solving the Telegraph Equation Model

Using the menu bar, select Solve > Solver Parameters. Once the Solver Parameters window appears, click the Time Stepping tab. Place a check mark in the Manual tuning of step size check box. Type 0.002 in the Initial time step edit field, as shown in Figure 3.41. Click OK.

Table 3.11 Initial Conditions Window

Initial Condition	Value
u1(t_0)	exp(–3*(x/0.2–1)^2)
u2(t_0)	0

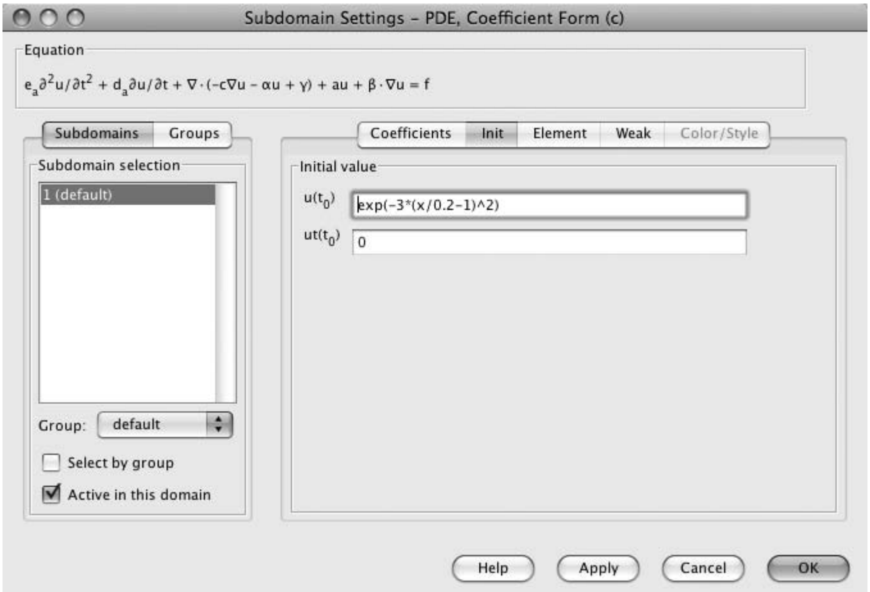


FIGURE 3.39 PDE window, Init page

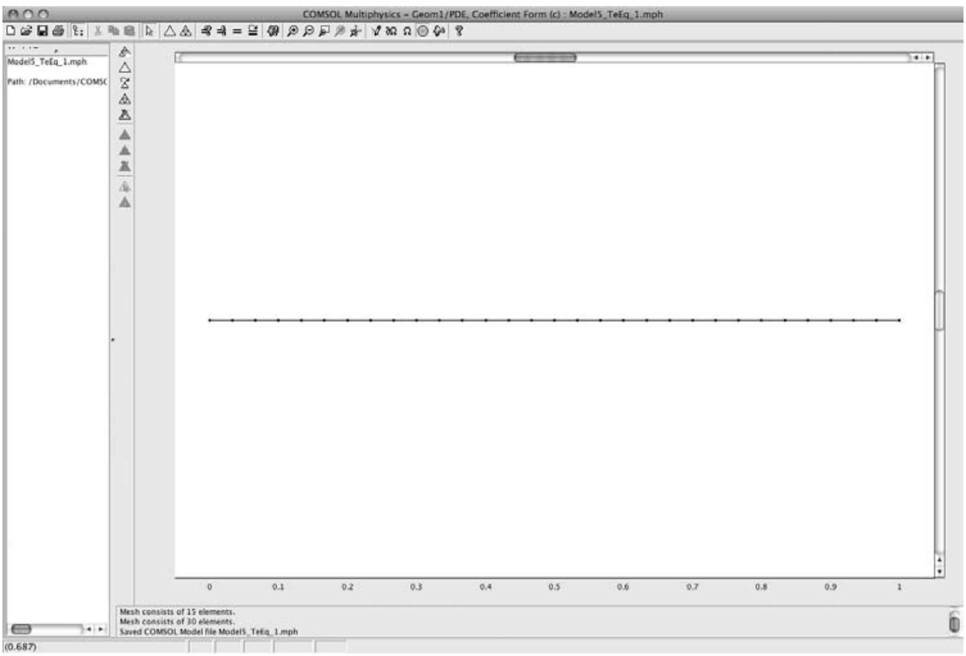


FIGURE 3.40 PDE, telegraph equation model mesh

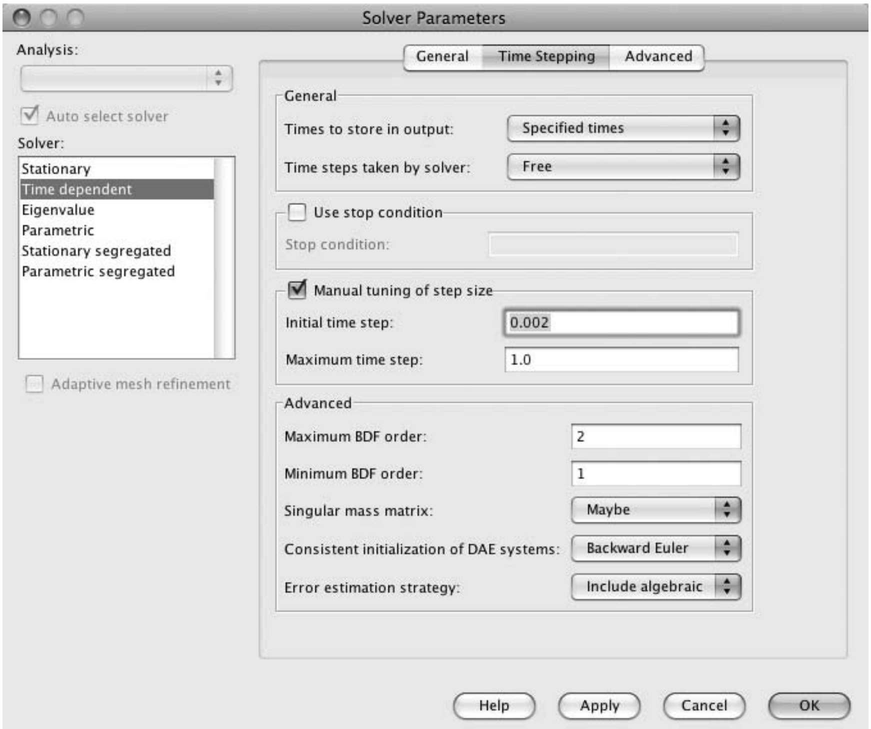


FIGURE 3.41 Solver Parameters window, Time Stepping page

NOTE The 0.002 time step is selected, in this case, to yield adequate solution resolution without requiring extensive resource consumption (computer/modeler time).

Using the menu bar, select Solve > Solve Problem. The solution for the final time interval is as shown in Figure 3.42.

Postprocessing

Using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the General tab. Place a check mark in the Keep current plot check box. Select “Solution at time: 0.” Click the Apply button. Select “Solution at time: 0.5.” Click the Apply button, and then click OK. Figure 3.43 shows the resulting plot of the pulse amplitude as it propagates from left to right.

Postprocessing Animation

This solution to the telegraph equation can also be viewed as an animation. To view this solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page,

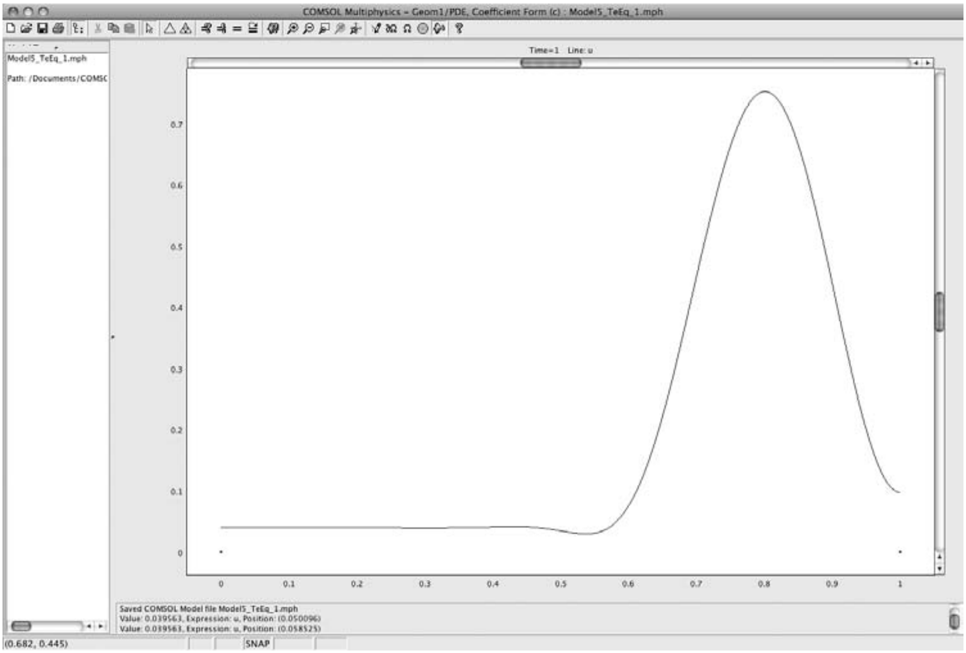


FIGURE 3.42 Telegraph equation model solution

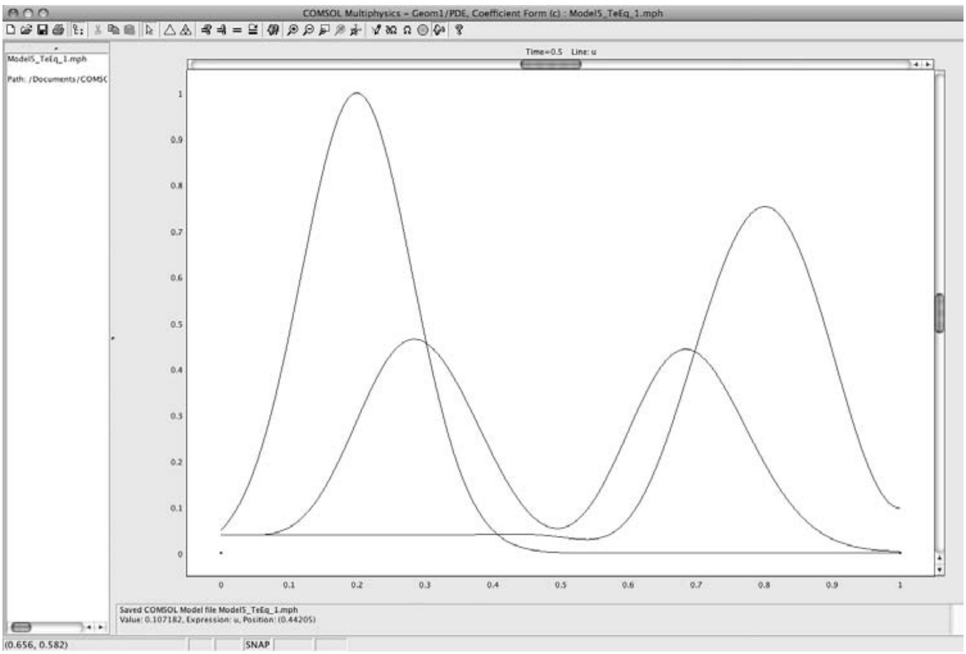


FIGURE 3.43 Telegraph equation pulse amplitude plot

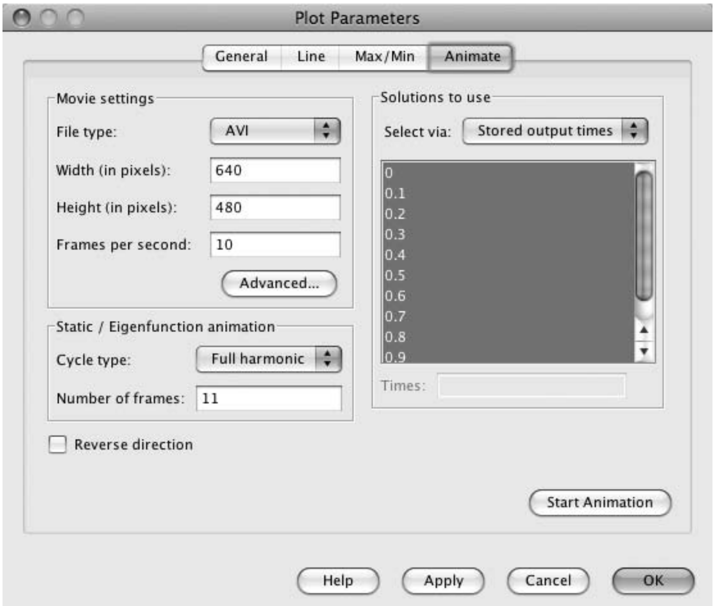


FIGURE 3.44 Telegraph Equation Plot Parameters window, Animate page

select all the solutions in the Stored output times window (see Figure 3.44). Click the Start Animation button. Save this telegraph equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3_TE_1.avi that was supplied with this book.

Select File > Save as. Type Model3_TeEq_1 in the Save As edit window.

First Variation on the Telegraph Equation Model

The previous solution to the telegraph equation shows a pulse propagating from left to right. Let us now explore how the model behaves when the initial conditions are modified. In this case, the argument is made smaller, reflecting the behavior of a lower-loss transmission line.

NOTE Information transmission relies on the measurement of differences, as stated earlier. To receive a message, the signal must be of detectable amplitude (analog), of detectable duration (time), and in the pass-band (correct frequency or frequency spread) of the receiver. The signal amplitude must be sufficiently greater than the detection threshold and above the noise level (on the average) to allow information to be collected.

First, save a new copy of the just-created telegraph equation model Model3_TeEq_1 as Model3_TeEq_2. You can then modify the telegraph equation model without being concerned about damaging the original model.

Table 3.12 Constants Window

Name	Expression
c	1
alpha	0.005
beta	0.005

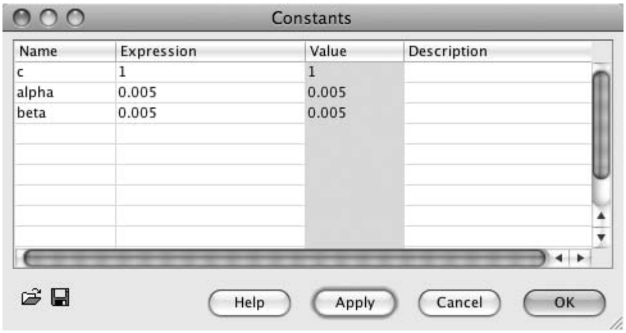


FIGURE 3.45 Constants window

Using the menu bar, select Options > Constants. After the Constants window appears, type the expressions indicated in Table 3.12 (also see Figure 3.45), and then click OK.

Boundary Conditions

Using the menu bar, select Physics > Boundary Settings. After the Boundary Settings window appears, select both boundaries 1 and 2. Enter or verify the settings indicated in Table 3.13 (also see Figure 3.46). Click OK.

Subdomain Settings

The next step in building the revised telegraph equation model is to set the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window

Table 3.13 Boundary Settings Window

Parameter	Boundary 1	Boundary 2	q	g
Type	Neumann	Neumann		
Setting			0	0

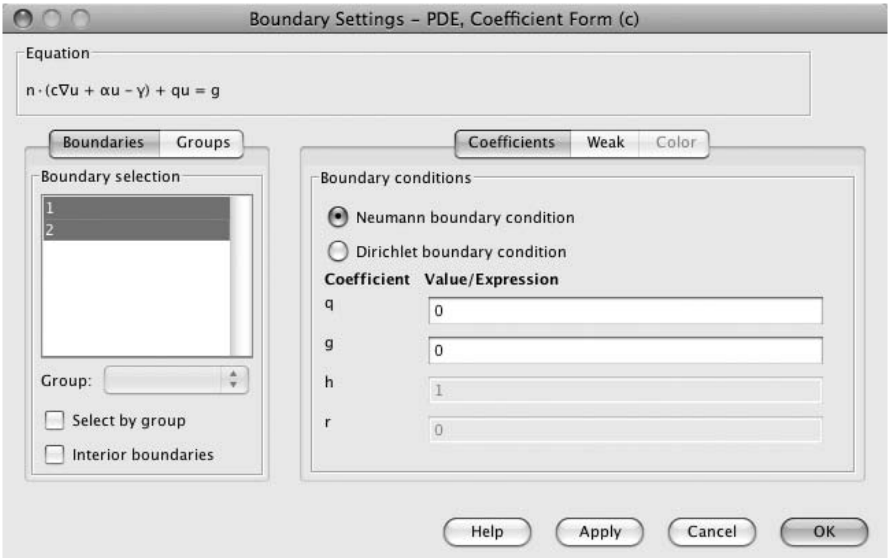


FIGURE 3.46 Boundary Settings window

appears, select subdomain 1. Enter or verify the coefficient values under the correct tab as shown in Table 3.14. Leave the other coefficient settings at their 0 value, as shown in Figure 3.47.

Once the PDE coefficients have been entered or verified, click the Init tab. Type or verify the initial conditions found in Table 3.15 in the edit windows, as shown in Figure 3.48. Click OK.

Model Reset

Select File > Reset Model > Yes.

Table 3.14 Subdomain Settings Window, PDE Coefficients

PDE Coefficient	Value
c	c*c
a	alpha*beta
f	–(alpha+beta)*ut
e_a	1
d_a	0

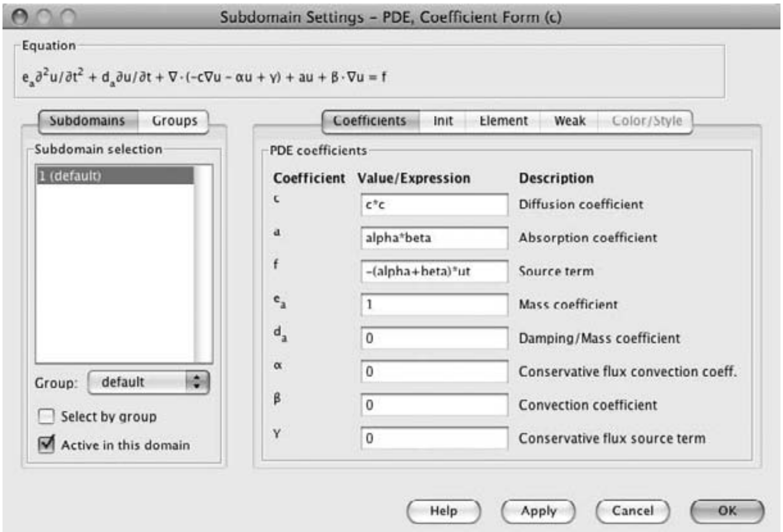


FIGURE 3.47 PDE, Subdomain Settings window, PDE Coefficients

Table 3.15 Initial Conditions Window

Initial Condition	Value
$u1(t_0)$	exp(−3*(x/0.2−1)^2)
$u2(t_0)$	0

Mesh Generation

Using the toolbar, select Initialize Mesh > Refine Mesh once. The final 30-element mesh is shown in Figure 3.49.

Solving the Telegraph Equation Model

Using the menu bar, select Solve > Solver Parameters. Once the Solver Parameters window appears, click the Time Stepping tab. Place a check mark in the Manual tuning of step size check box. Type 0.002 in the Initial time step edit field, as shown in Figure 3.50. Click OK.

NOTE As mentioned in an earlier note, the 0.002 time step is selected to yield adequate solution resolution without requiring extensive resource consumption (computer/modeler time).

Using the menu bar, select Solve > Solve Problem.

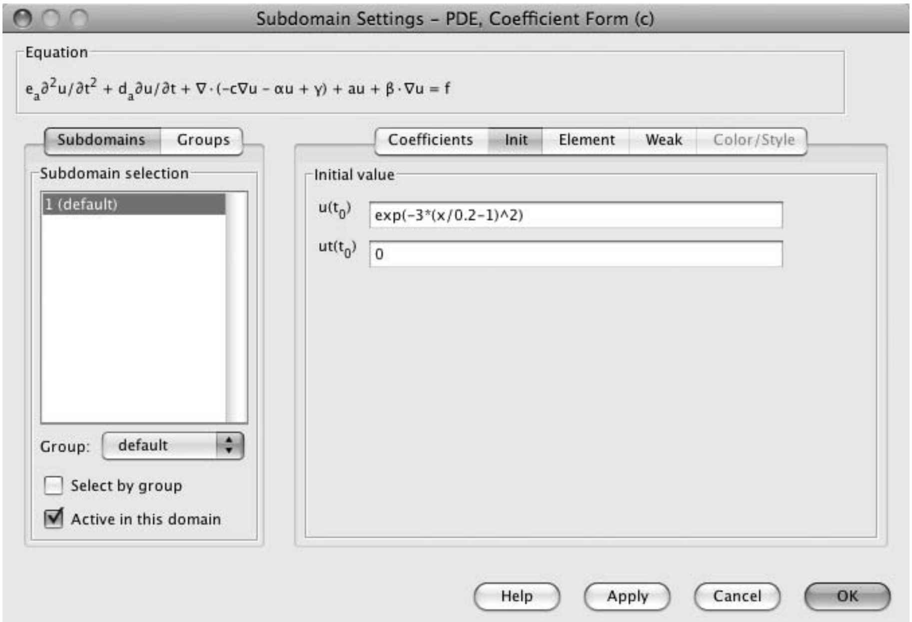


FIGURE 3.48 PDE, Coefficient window, Init page

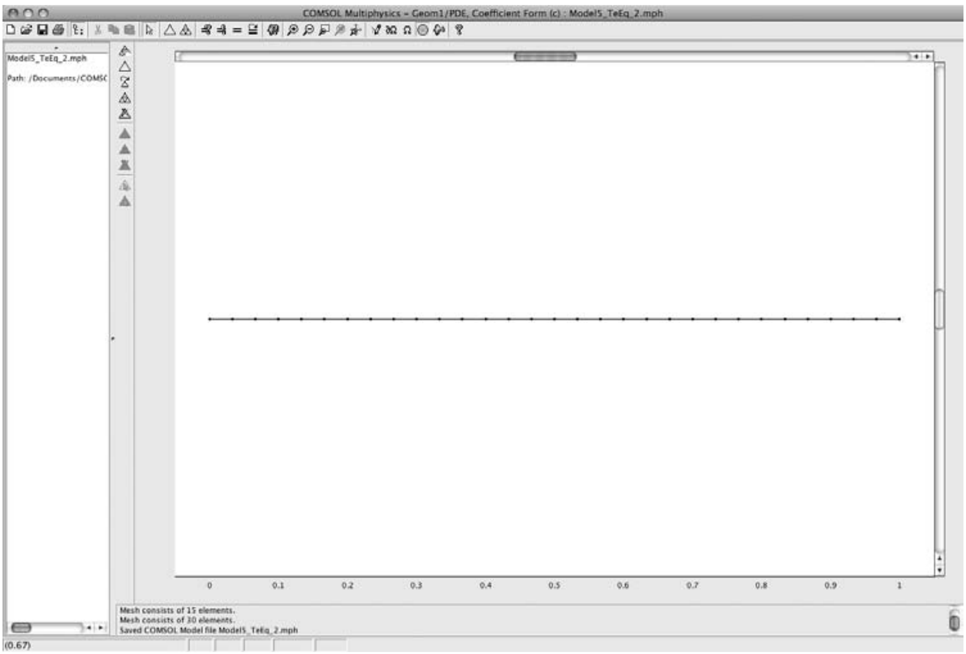


FIGURE 3.49 Telegraph equation model mesh

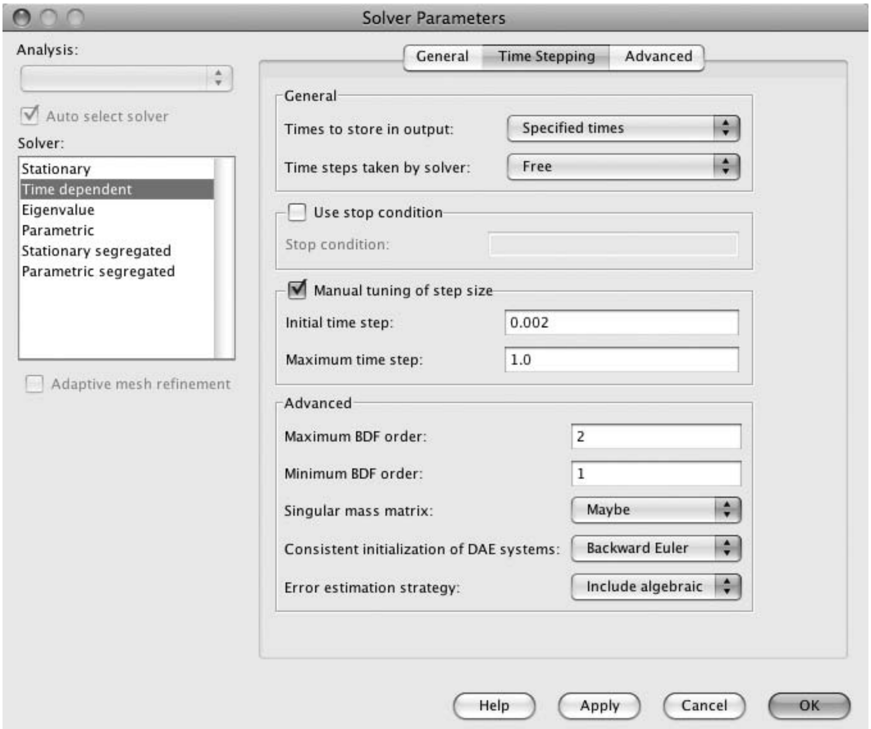


FIGURE 3.50 Solver Parameters window, Time Stepping page

Postprocessing

Using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the General tab. Place a check mark in the Keep current plot check box. Select "Solution at time: 0." Click the Apply button. Select "Solution at time: 0.5." Click the Apply button, and then click OK. Figure 3.51 shows the resulting plot of the pulse amplitude as it propagates from left to right. Note that the final pulse amplitude is 0.9 as compared to 0.7 for the original model.

Postprocessing Animation

This solution to the telegraph equation can also be viewed as an animation. To view this solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 3.52). Click the Start Animation button. Save this telegraph equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3_TE_2.avi that was supplied with this book.

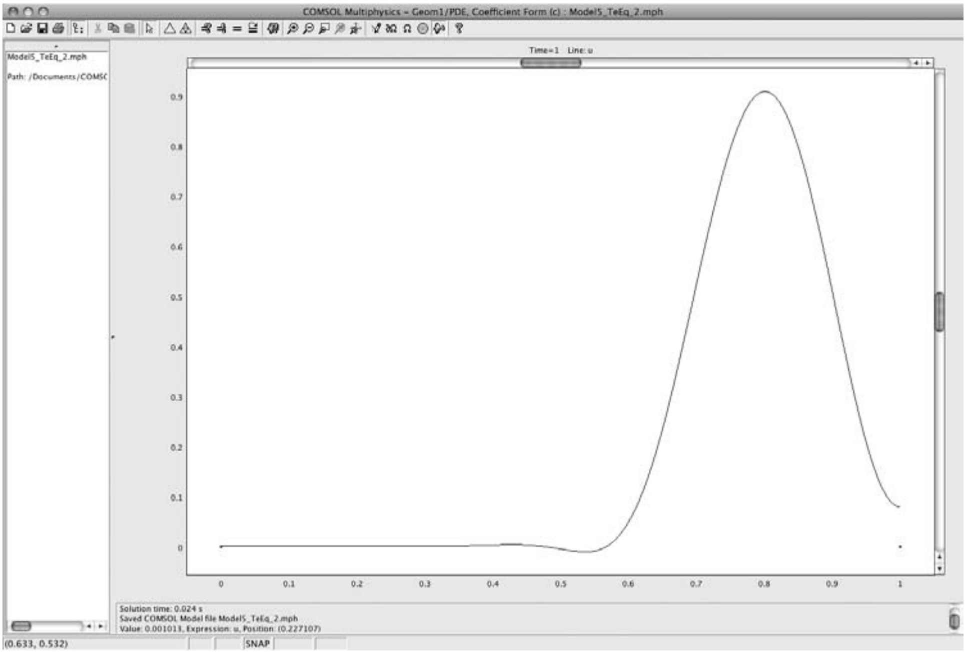


FIGURE 3.51 Telegraph equation pulse amplitude plot, low-loss line

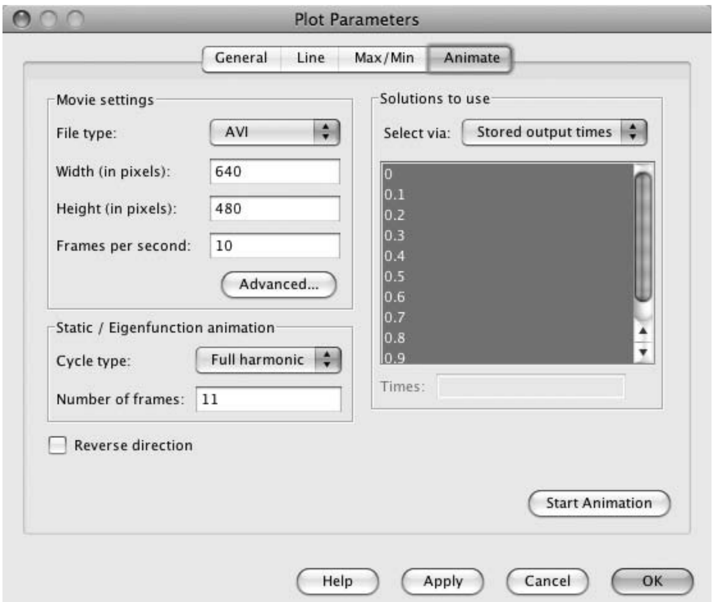


FIGURE 3.52 Telegraph equation Plot Parameters window, Animate page

Table 3.16 Constants Window

Name	Expression
c	1
alpha	5
beta	5

Select File > Save as. Type Model3_TeEq_2 in the Save As edit window. Click the Yes button to replace the earlier file.

Second Variation on the Telegraph Equation Model

The previous solution to the telegraph equation shows a pulse propagating from left to right. Let us now explore how the model behaves when the initial conditions are modified. In this case, the argument is made larger, reflecting the behavior of a higher-loss transmission line.

NOTE As stated earlier, information transmission relies on the measurement of differences. To receive a message, the signal must be of detectable amplitude (analog), of detectable duration (time), and in the pass-band (correct frequency or frequency spread) of the receiver. The signal amplitude must be sufficiently greater than the detection threshold and above the noise level (on the average) to allow information to be collected.

First, save a new copy of the just-created telegraph equation model Model3_TeEq_2 as Model3_TeEq_3. You can then modify the telegraph equation model without being concerned about damaging the just-built model.

Using the menu bar, select Options > Constants. After the Constants window appears, type the expressions indicated in Table 3.16 (also see Figure 3.53), and then click OK.

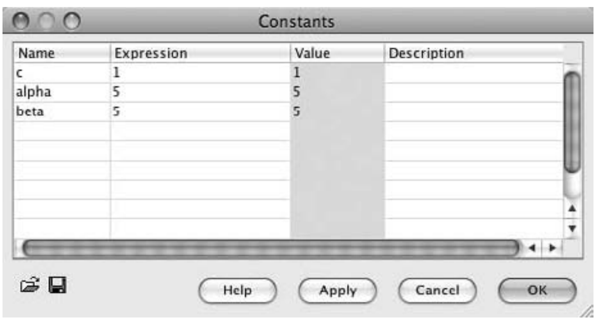


FIGURE 3.53 Constants window

Table 3.17 Boundary Settings Window

Parameter	Boundary 1	Boundary 2	q	g
Type	Neumann	Neumann		
Setting			0	0

Boundary Conditions

Using the menu bar, select Physics > Boundary Settings. After the Boundary Settings window appears, select both boundaries 1 and 2. Enter or verify the settings indicated in Table 3.17, as shown in Figure 3.54. Click OK.

Subdomain Settings

The next step in building the revised telegraph equation model is to set the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select subdomain 1. Enter or verify the coefficient values under the correct tab as shown in Table 3.18. Leave the other coefficient settings at their 0 value, as shown in Figure 3.55.

Once the PDE coefficients have been entered or verified, click the Init tab. Type or verify the initial conditions found in Table 3.19 in the edit windows, as shown in Figure 3.56. Click OK.

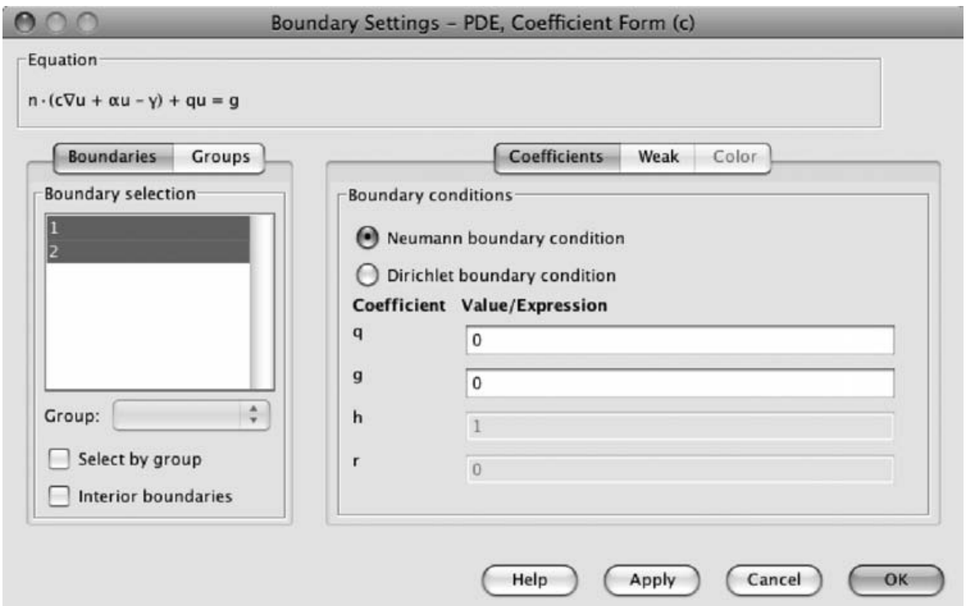


FIGURE 3.54 Boundary Settings window

Table 3.18 Subdomain Settings Window, PDE Coefficients

PDE Coefficient	Value
c	c*c
a	alpha*beta
f	–(alpha+beta)*ut
e_a	1
d_a	0

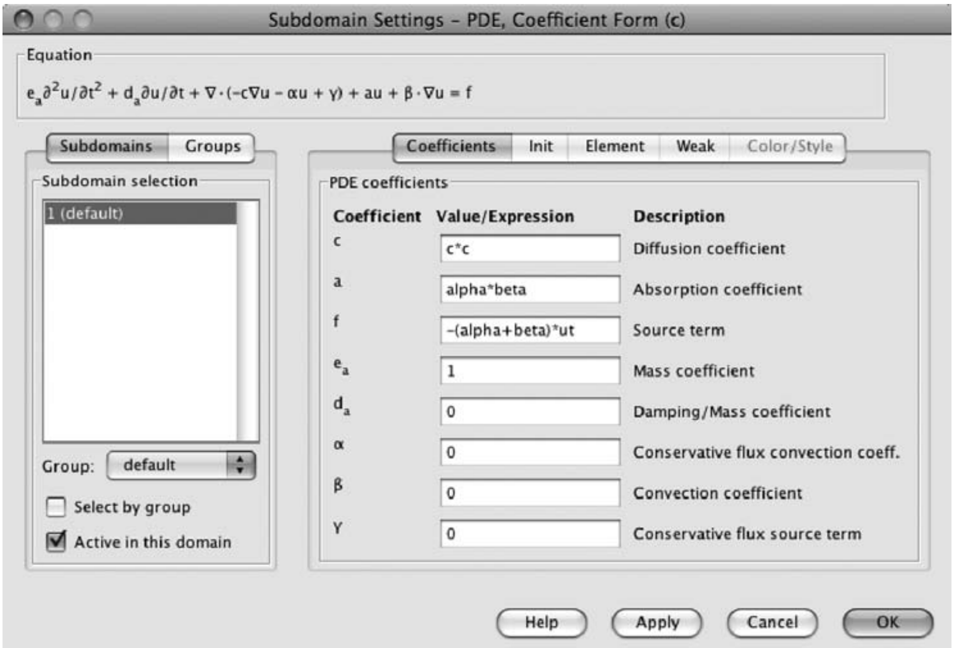


FIGURE 3.55 PDE, Subdomain Settings window, PDE Coefficients

Table 3.19 Initial Conditions Window

Initial Condition	Value
$u1(t_0)$	exp(–3*(x/0.2–1)^2)
$u2(t_0)$	0

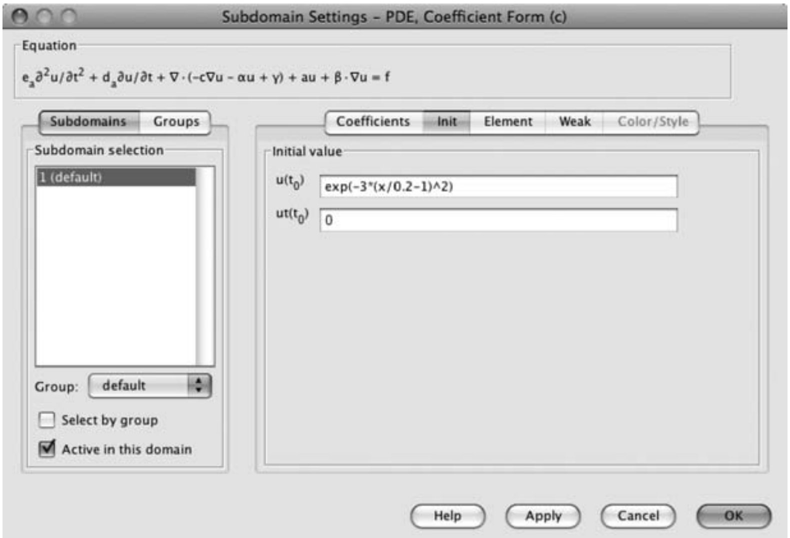


FIGURE 3.56 PDE, Coefficient window, Init page

Model Reset

Select File > Reset Model > Yes.

Mesh Generation

Using the tool bar, select Initialize Mesh > Refine Mesh once. The final 30-element mesh is shown in Figure 3.57.

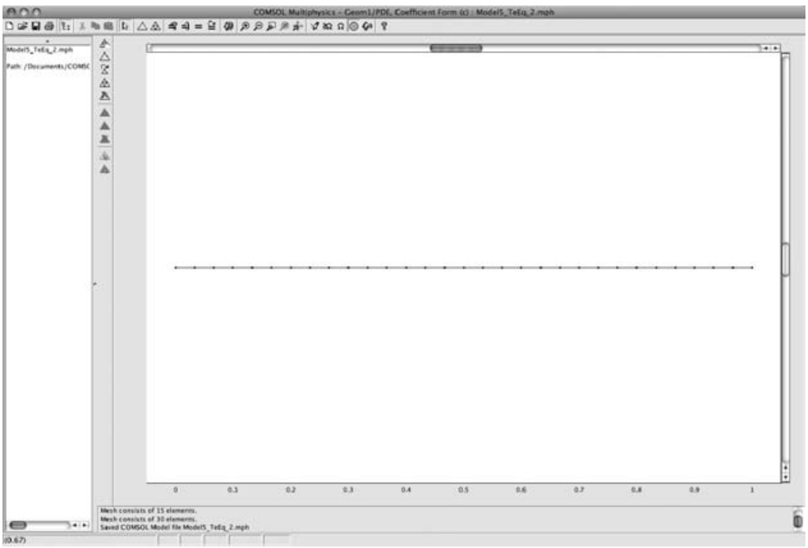


FIGURE 3.57 Telegraph equation model mesh

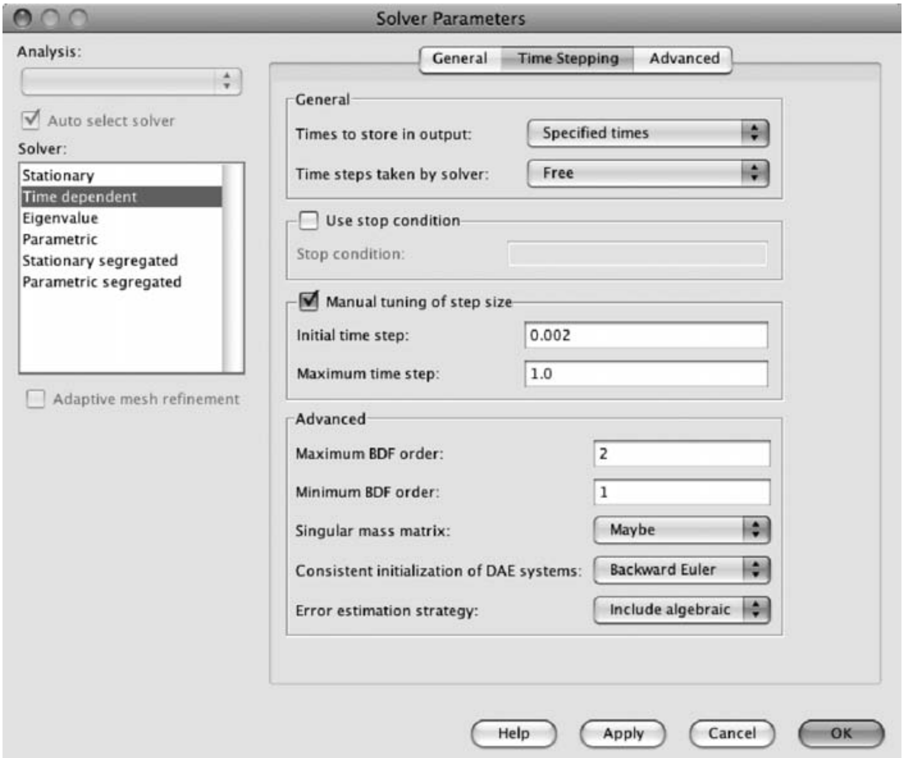


FIGURE 3.58 Solver Parameters window, Time Stepping page

Solving the Telegraph Equation Model

Using the menu bar, select Solve > Solver Parameters. Once the Solver Parameters window appears, click the Time Stepping tab. Place a check mark in the Manual tuning of step size check box. Enter or verify 0.002 in the Initial time step edit field, as shown in Figure 3.58. Click OK.

NOTE As mentioned in an earlier note, the 0.002 time step is selected to yield adequate solution resolution without requiring extensive resource consumption (computer/modeler time).

Using the menu bar, select Solve > Solve Problem. The solution for the final time interval is as shown in Figure 3.59.

Postprocessing

Using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the General tab. Place a check mark in the Keep

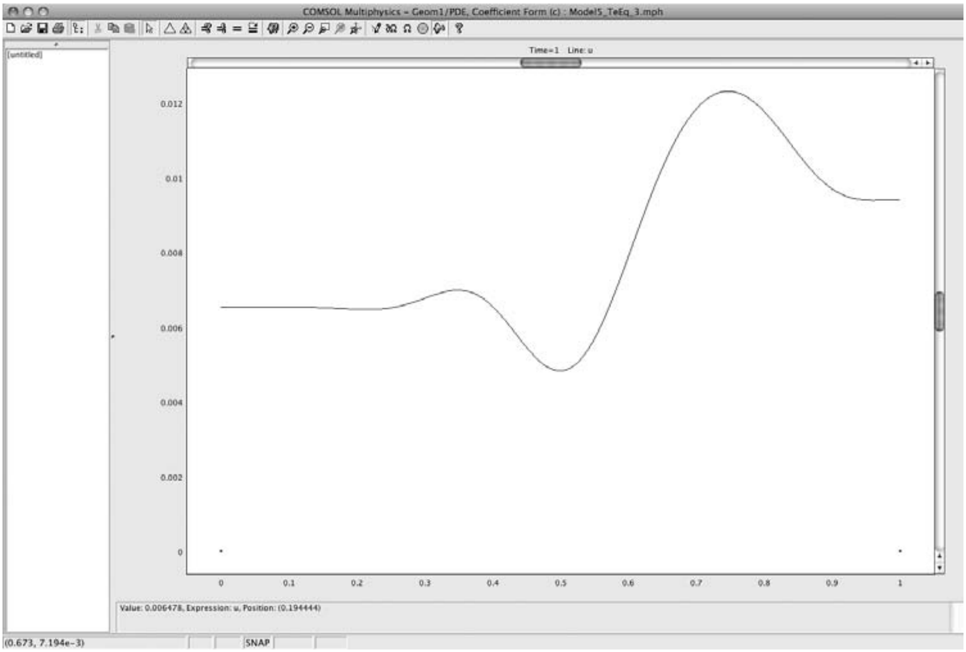


FIGURE 3.59 Telegraph equation solution, final time interval

current plot check box. Select "Solution at time: 0." Click the Apply button. Select "Solution at time: 0.5." Click the Apply button, and then click OK. Figure 3.60 shows the resulting plot of the pulse amplitude as it propagates from left to right.

Postprocessing Animation

This solution to the telegraph equation can also be viewed as an animation. To view this solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 3.61). Click the Start Animation button. Save this telegraph equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3_TE_3.avi that was supplied with this book.

Select File > Save as. Type Model3_TeEq_3 in the Save As edit window. Click the Yes button to replace the earlier file.

1D Telegraph Equation Models: Summary and Conclusions

The telegraph equation is a powerful tool that can be used to model wave propagation in diverse transmission lines. It can be used to thoroughly characterize the propagation conditions of coaxial lines, twin pair lines, microstrip lines, and more. The telegraph equation is easily and simply modeled with a 1D PDE mode model.

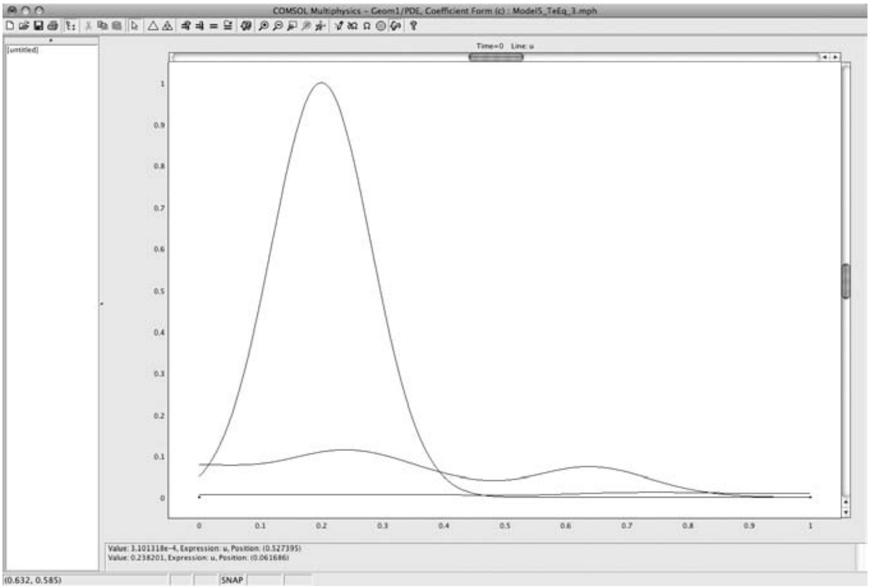


FIGURE 3.60 Telegraph equation pulse amplitude plot, high-loss line

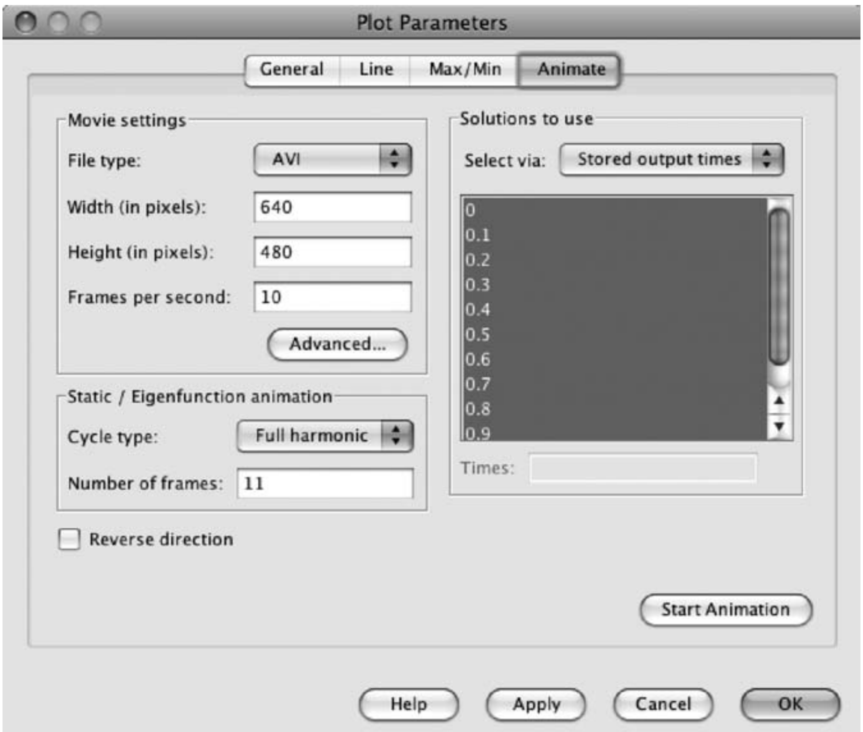


FIGURE 3.61 Telegraph equation Plot Parameters window, Animate page

References

1. http://en.wikipedia.org/wiki/KdV
2. http://en.wikipedia.org/wiki/List_of_nonlinear_partial_differential_equations
3. http://en.wikipedia.org/wiki/Exactly_solvable
4. http://en.wikipedia.org/wiki/Nonlinear_system
5. http://en.wikipedia.org/wiki/John_Scott_Russell
6. http://en.wikipedia.org/wiki/Soliton
7. http://en.wikipedia.org/wiki/Soliton_%28optics%29
8. http://en.wikipedia.org/wiki/Signal_to_noise
9. http://en.wikipedia.org/wiki/Soliton_%28optics%29
10. http://en.wikipedia.org/wiki/Telegraph_equation
11. http://en.wikipedia.org/wiki/Oliver_Heaviside
12. http://en.wikipedia.org/wiki/Transmission_line

Exercises

1. Build, mesh, and solve the 1D KdV equation problem presented in this chapter.
2. Build, mesh, and solve the first variation of the KdV equation problem presented in this chapter.
3. Build, mesh, and solve the second variation of the KdV equation problem presented in this chapter.
4. Build, mesh, and solve the telegraph equation problem presented in this chapter.
5. Build, mesh, and solve the first variation of the telegraph equation problem presented in this chapter.
6. Build, mesh, and solve the second variation of the telegraph equation problem presented in this chapter.
7. Explore other variations of the arguments in the KdV equation model.
8. Explore other variations of the arguments in the telegraph equation model.
9. Explore the role that characteristic impedance plays in transmission lines.

4 2D Modeling

In This Chapter

■ 2D Guidelines for New COMSOL® Multiphysics® Modelers

2D Modeling Considerations

2D modeling can be less difficult than 1D modeling, having fewer implicit assumptions, and yet potentially can still be a challenging type of model to build, depending on the underlying physics involved, irrespective of the modeling software utilized. The least difficult aspect of 2D model building arises from the fact that the geometry is relatively simple: In a 2D model, the modeler has only a single plane as the modeling space. However, the physics in a 2D model can range from relatively easy to extremely complex.

NOTE COMSOL® Multiphysics® software has two 2D modeling modes: 2D (beginning-level through advanced-level 2D modeling) and 2D Axisymmetric (advanced-level 2D modeling). In keeping with the introductory focus of the material in this text, both model types, their associated physics, and the related methodology for

use of the models, are introduced in Chapters 4 and 5. Significantly more advanced 2D modeling techniques exist than are presented in these two chapters. Examples of some of those more challenging techniques are reserved for introduction in Chapters 6 and 7. For further expansion of the 2D modeling horizons, refer to the COMSOL manuals, the COMSOL Website, and the general COMSOL Multiphysics software-related research literature.

The 2D model implicitly assumes, in compliance with the laws of physics, that the energy flow, the materials properties, the environment, and any other conditions and variables that are of interest are homogeneous, isotropic, and constant, unless otherwise specified, throughout the entire domain of interest, both within the model and, through the boundary conditions, in the environs of the model. Bearing that in mind, the modeler needs to ensure that all of the modeling conditions and associated parameters (default settings) in each new model created have been properly considered, defined, or set to the appropriate values.

The modeler also needs to seriously consider the steps that will be required in properly establishing the correct postprocessing and visualization settings to extract the desired information from the modeling solution. The default parameter settings on any given model will probably not present exactly the information that the modeler needs or desires, although it will probably come close. It is the responsibility of the modeler to determine exactly which of the myriad of postprocessing and visualization choices available in the COMSOL Multiphysics software to employ.

NOTE It is always preferable for the modeler to be able to accurately anticipate the expected behavior (results) of the model and the way in which those results should be presented. Never assume that the default values that are initially present when the model is first created will suit the needs of a new model. Always verify that the values employed in the model are the correct values needed for that model. Calculated solution values that significantly deviate from the expected values or from comparison values measured in experimentally derived realistic models are probably indicative of one or more modeling errors either in the original model design, in the earlier model analysis, or in the understanding of the underlying physics, or are simply due to human error.

Coordinate System

In 2D models, there are three coordinates: space (x), space (y), and time (t). In a steady-state solution to a 2D model, parameters can vary only as a function of position in the space (x) and space (y) coordinates. Such a 2D model represents the parametric condition of the model in a time-independent mode (quasi-static). In a transient solution model, parameters can vary both by position in space (x) and space (y), and in time (t).

The transient solution model is essentially a sequential collection of steady-state (quasi-static) solutions. The space coordinates (x) and (y) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values ($x_{min} <= x <= x_{max}$) and ($y_{min} <= y <= y_{max}$). The time coordinate (t) represents the range of values ($t_{min} <= t <= t_{max}$) from the beginning of observation period (t_{min}) to the end of observation period (t_{max}).

To assist the reader to achieve a broader exposure to the applicability of the physics discussed here and to demonstrate the power of the basic COMSOL 2D modeling techniques, the modeling examples in this chapter illustrate techniques from two substantially different, but important and widely applied technologies currently employed in applied engineering and physics. The first example presented, electropolishing, explores the modeling of a processing methodology utilized in the fabrication and finishing of many metallic objects that require a smooth surface (e.g., microscope samples, precision metal parts, medical equipment and tools, large and small metal drums, thin analytical samples, vacuum chambers). The second example, the Hall effect (a magnetic sensor technology), explores the behavior of currents (electrons or holes) flowing in a semiconducting material (e.g., Si, Ge) under the influence of an external magnetic field.

2D Electrochemical Polishing (Electropolishing) Theory

Electrochemical polishing[1] (also known as electropolishing[2]) is a well-known process in the metal finishing industry. It allows the finished surface smoothness of a conducting material to be cleanly controlled to a high degree of precision, using relatively simple processing equipment. The electrochemical polishing technique eliminates the abrasive residue typically present on the polished surface from a mechanical polishing process; it also eliminates the need for complex, mechanical polishing machinery.

NOTE The science of electricity, and consequently that of electrochemistry, started with the work of William Gilbert through his study of magnetism. Gilbert first published his studies in 1600.[3] Charles-Augustin de Coulomb,[4] Joseph Priestley,[5] Georg Ohm,[6] and others made additional independent contributions that furthered the basic understanding of the nature of electricity and electrochemistry. Those contributions led to the discovery and disclosure by Michael Faraday[7] of his two laws of electrochemistry in 1832.

The numerical solution model for electrochemical polishing was originally developed by COMSOL for distribution with the Multiphysics software as a COMSOL Multiphysics electromagnetics model. This model introduces two important basic concepts, the first in applied physics and the second in applied

modeling: (1) electropolishing and (2) the moving mesh (ALE = arbitrary Lagrangian–Eulerian[8]). The electrochemical polishing model built in this chapter is substantially the same as presented in the COMSOL Model Library. In this chapter, following development of the first model, variations and expansions on the basic electrochemical polishing model are explored.

NOTE It is important for the new modeler to personally build each model presented within this text. There is no substitute in the path to an understanding of the modeling process for the hands-on experience of actually building, meshing, solving, and postprocessing a model. Many times the inexperienced modeler will make and subsequently correct errors, adding to his or her experience and fund of modeling knowledge. Even building the simplest model will expand the modeler's fund of knowledge.

Polishing (smoothing) of a material surface, via either mechanical or electrochemical means, results from the reduction of asperities (bumps) to achieve a nominally smooth surface (uniform thickness $\pm \triangle$ thickness). In a mechanical polishing technique, the reduction of asperities occurs through the use of finer (smaller) and finer grit (abrasive) sizes. The mechanical polishing of many surfaces is difficult, if not impossible, owing to the complexity and/or physical size of such surfaces. Figure 4.1 shows a simple asperity, as will be modeled in this section of the chapter.

The surface of the electrode, using this method, is polished by the differential removal of material from local asperities in selected areas, accomplished through the immersion of the nominally rough electrode in an electrolyte and the application of a current (electron bombardment). A first-order approximation to the experimentally observed material removal process is that the rate (velocity) of material removal (U) from the electrode surface is proportional to the amplitude of the current and direction of the current $\mathbf{J}$, relative to the local surface normal vector $\mathbf{n}$ (see Figure 4.2):

$$U = -K^*\mathbf{J} \cdot \mathbf{n} = -K^*J_n \tag{4.1}$$

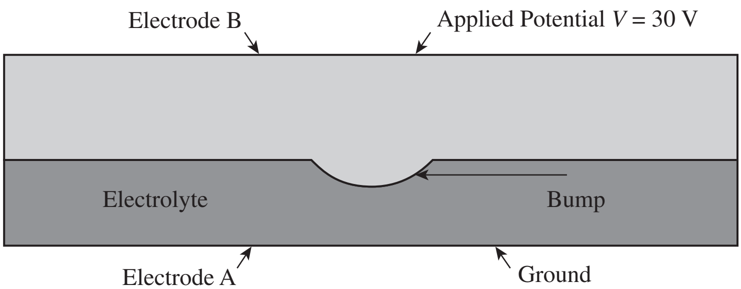


FIGURE 4.1 2D An asperity (bump) on an electrode

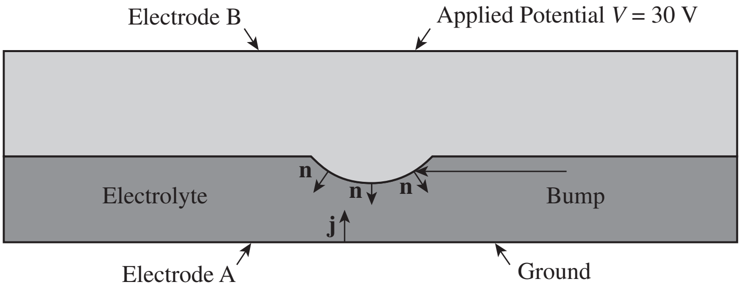


FIGURE 4.2 Surface normal vector **n** and the current vector **J**

NOTE The electropolishing technique, to a first approximation, is the inverse of electroplating. As a result, the rate of removal of material (velocity = U) from the nominally rough surface of the positive electrode is proportional to the normal current density at the positive electrode surface, as shown in equation 4.1.

The exact value of the proportionality constant (K) in physical applications (e.g., research experiments, processing) is determined by the electrode material, the electrolyte, the temperature, and other factors, and, to some extent, will be explored in later examples in this chapter.

For this model, the proportionality constant is chosen to be

$$K = 1.0 \times 10^{-11}\,\mathrm{m}^3/(\mathrm{A}^*\mathrm{s}) \tag{4.2}$$

where m = meters
A = amperes
s = seconds

Obviously, because material is removed from the positive electrode during the electropolishing process, the spacing between the upper and lower electrodes will increase. The time rate of change of the model geometry (electrode spacing) needs to be accommodated somewhere within the model. The Moving Mesh (ALE = arbitrary Lagrangian–Eulerian) Application Mode accommodates that time rate of change, resulting from the normal current (J_n) flowing in the electrolyte during the quasi-static use of the Conductive Media DC Application Mode.

NOTE The Moving Mesh Application Mode allows the modeler to create models in which the physics of the process introduces and controls geometric changes in the model. However, the modeler must know and work carefully within the limits of the modeling system. The Moving Mesh Application Mode is a powerful tool. However, the calculated mesh parameters can drift, as the mesh is deformed and ultimately lead to

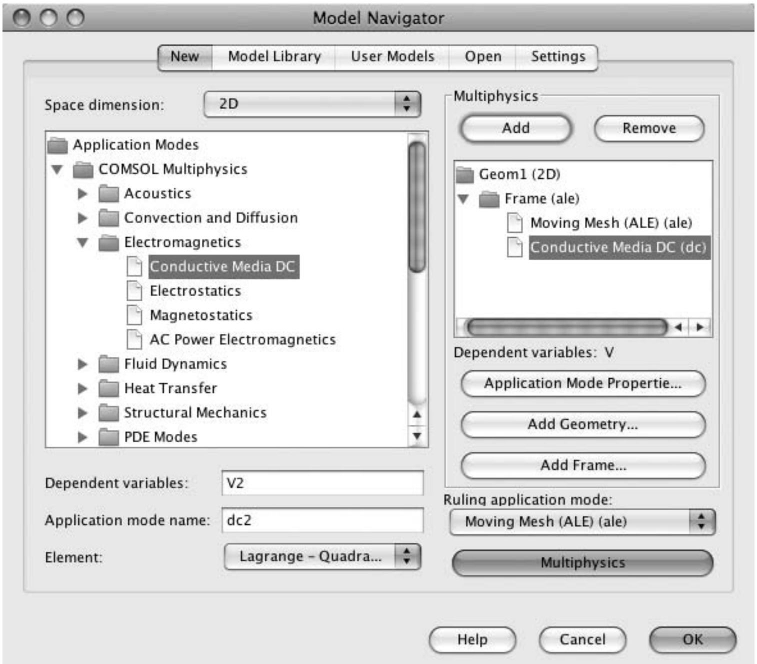


FIGURE 4.3 2D Electropolishing_1 Model Navigator setup

nonphysical, nonconvergent results. Avoidance of such nonphysical results requires the modeler to understand the basic physics of the modeled problem and to choose the meshing method that yields the best overall results.

COMSOL 2D Electrochemical Polishing Model

To start building the Electropolishing_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select “2D” from the Space dimension pull-down list. Select COMSOL Multiphysics > Deformed Mesh > Moving Mesh (ALE) > Transient analysis. Click the Multiphysics button, and then click the Add button.

From the Application Modes list, select COMSOL Multiphysics > Electromagnetics > Conductive Media DC. Click the Add button. See Figure 4.3. Click OK.

Table 4.1 Constants Edit Window

Name	Expression	Description
K	1.0e-11[m^3/(A*s)]	Coefficient of proportionality

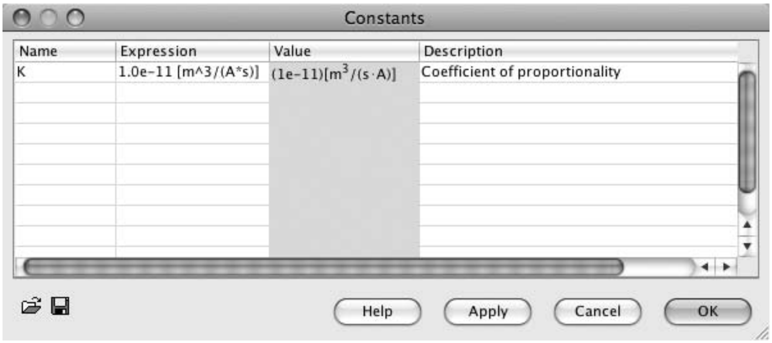


FIGURE 4.4 2D Electropolishing_1 model Constants edit window

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.1; see Figure 4.4. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 2.8 and a height of 0.4. Select "Base: Corner" and set X equal to −1.4 and Y equal to 0 in the Rectangle edit window. See Figure 4.5.

Click the Apply button, and then click OK. See Figure 4.6.

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.3. Select "Base: Center" and set X equal to 0 and Y equal to 0.6 in the Circle edit window. See Figure 4.7.

Click the Apply button, and then click OK. See Figure 4.8.

Select both the rectangle and the circle by clicking on the rectangle and Shift-clicking on the circle. See Figure 4.9.

Click the Difference button on the Draw toolbar to remove the overlapping portion of the circle from the rectangle. The upper surface of the electrolyte rectangle

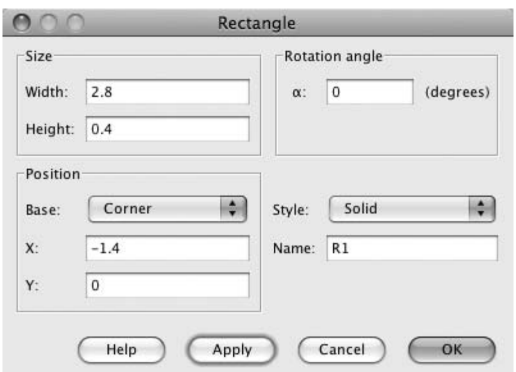


FIGURE 4.5 2D Electropolishing_1 model Rectangle edit window

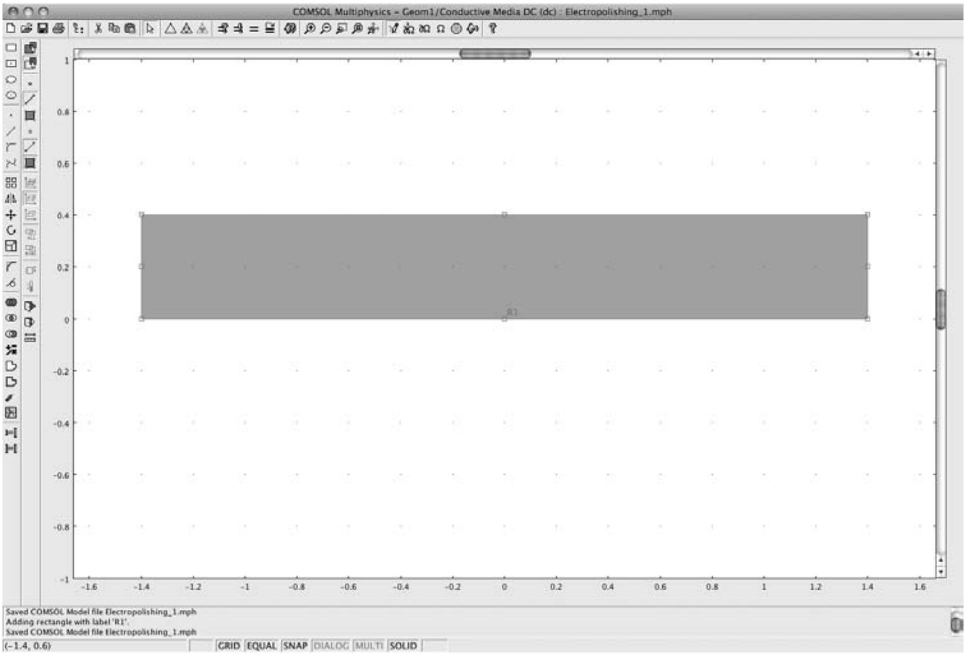


FIGURE 4.6 2D Electropolishing_1 model electrolyte rectangle

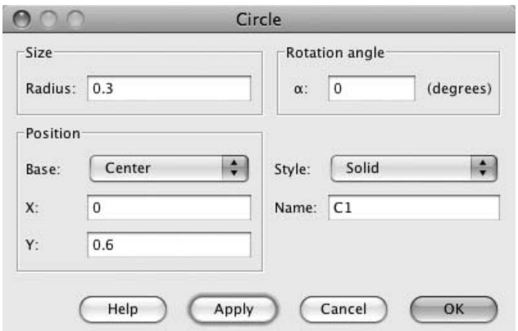


FIGURE 4.7 2D Electropolishing_1 model Circle edit window

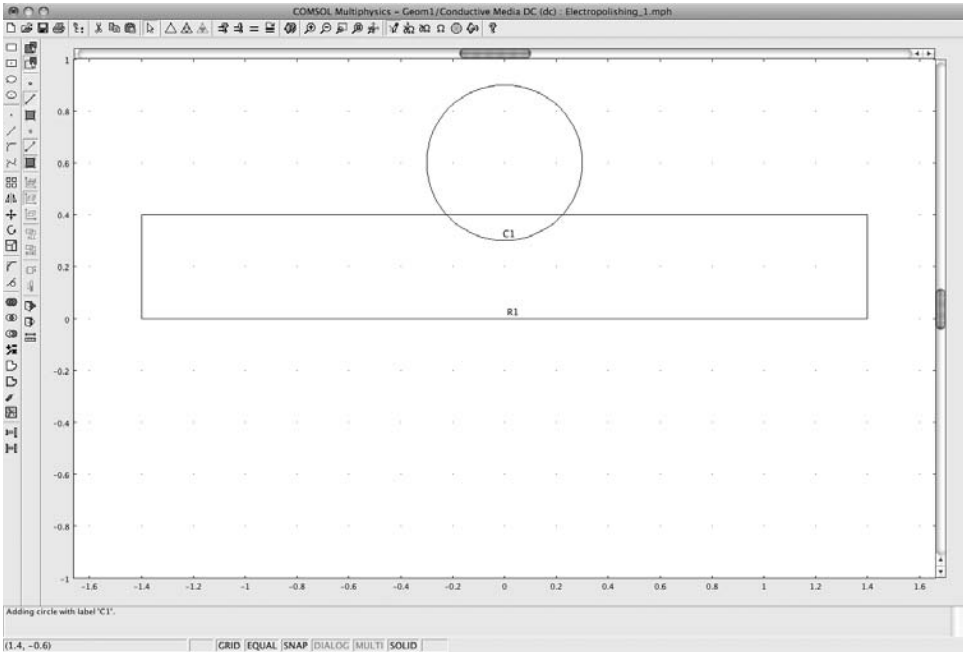


FIGURE 4.8 2D Electropolishing_1 model rectangle and circle

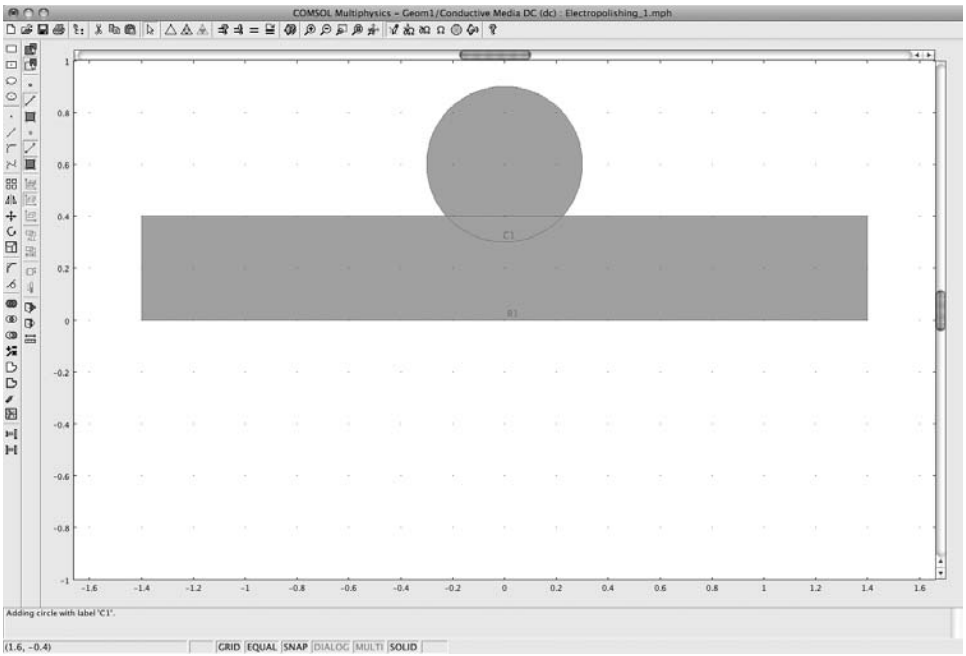


FIGURE 4.9 2D Electropolishing_1 model selected rectangle and circle

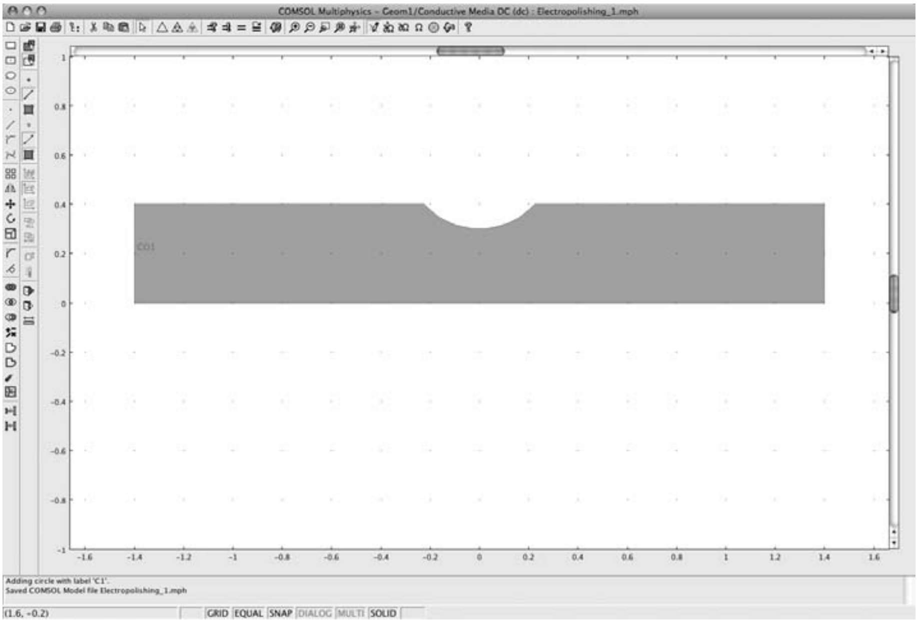


FIGURE 4.10 2D Electropolishing_1 model electrode with asperity

(CO1) is the lower surface of the electrode, with the asperity, that will be electropolished. See Figure 4.10.

NOTE The model geometry, as presently scaled, is 1.4 meters in length and 0.4 meter in height. Electropolishing is typically applied as the final finishing (smoothing) step in a precision fabrication process (e.g., metallographic samples, vacuum chambers). Thus the model geometry will need to be reduced in scale to emulate reality.

Click on the text “CO1.” Next, click the Scale button on the Draw toolbar. Enter 1e-3 in both the X and Y Scale factor edit windows. See Figure 4.11. Click OK.

Click the Zoom Extents button on the menu bar. See Figure 4.12.

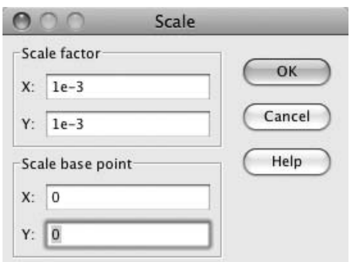


FIGURE 4.11 2D Electropolishing_1 model Scale edit window

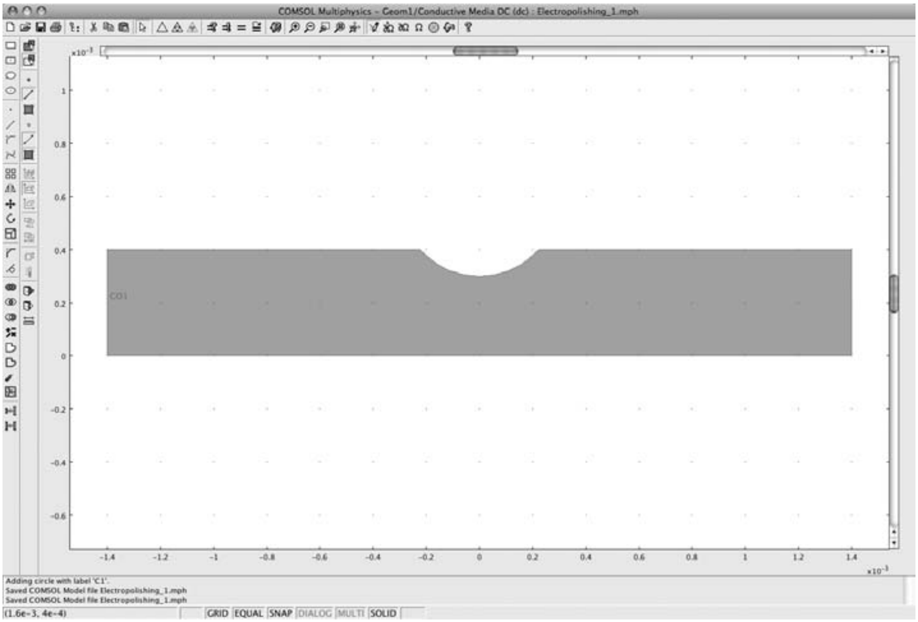


FIGURE 4.12 2D Electropolishing_1 model scaled electrolyte/electrode geometry

Physics Subdomain Settings: Conductive Media DC

Having established the 2D geometry for the electrochemical polishing model (a rectangle with a negative asperity on the upper surface), the next step is to define the fundamental physics conditions. Using the menu bar, select Multiphysics > Conductive Media DC. Next, using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain).

Enter 10 in the Electric conductivity (σ) edit window. See Figure 4.13. Click OK.

Physics Boundary Settings: Conductive Media DC

Using the menu bar, Select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value as shown in Table 4.2, and then click OK. See Figure 4.14.

Table 4.2 Subdomain Settings, Conductive Media DC Window

Boundary	Boundary Condition	Value/Expression
1, 5	Electric insulation	—
3, 4, 6 ,7	Electric potential	30
2	Ground	—

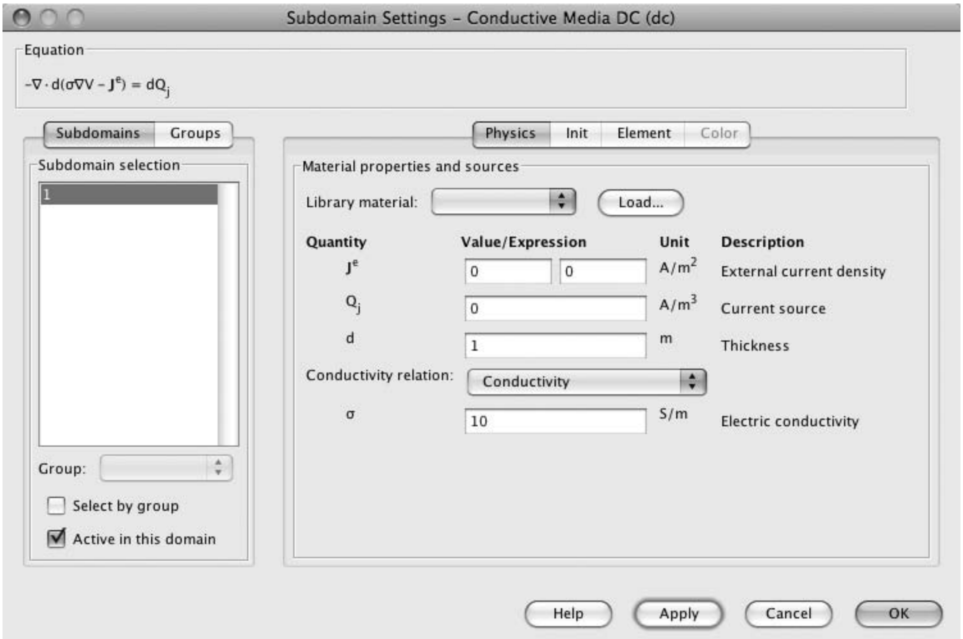


FIGURE 4.13 Subdomain Settings window

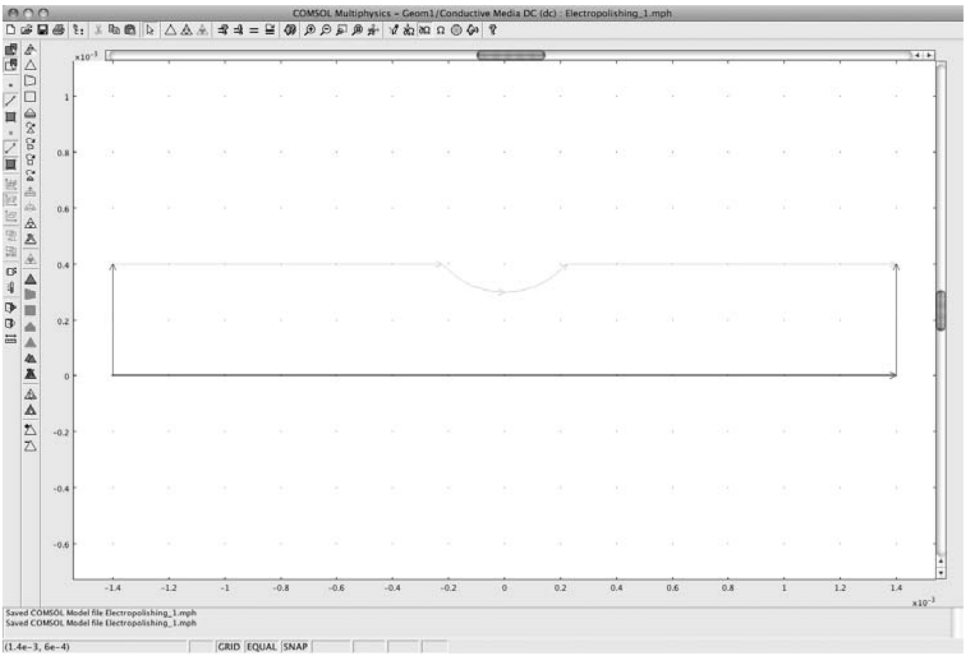


FIGURE 4.14 2D Electropolishing_1 model Boundary Settings, Conductive Media DC: boundaries set

Table 4.3 Boundary Settings, Moving Mesh (ALE) Window

Boundary	Coordinate	Boundary Condition	Value/Expression
1, 5	Global	Mesh velocity	vx = 0
3, 4, 6, 7	Tangent and normal Deformed mesh	Mesh velocity	vn = −K*nJ_dc
2	Global	Mesh displacement	dx = 0, dy = 0

Physics Boundary Settings: Moving Mesh (ALE)

Using the menu bar, select Multiphysics > Moving Mesh (ALE). Next, using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value as shown in Table 4.3, and then click OK. See Figure 4.15.

Mesh Generation

On the menu bar, click the Initialize Mesh button once.

Click the Refine Mesh button once. This results in a mesh of approximately 700 elements.

Click OK. See Figure 4.16.

Solving the 2D Electrochemical Polishing Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver. Enter 0:1:10 (typical values) in the Times edit window, as shown in Figure 4.17. This instruction causes the Solver to divide the modeling time-space into 10 equal intervals, over the period from 0 to 10 seconds. Click the Apply button, and then click OK.

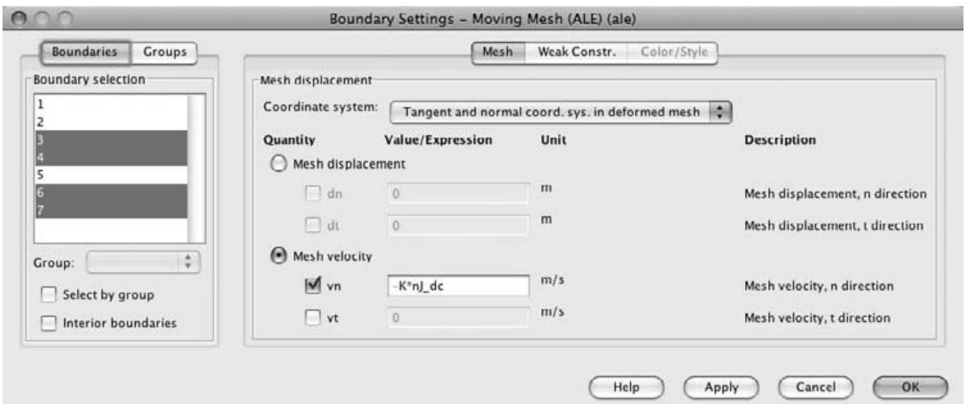


FIGURE 4.15 2D Electropolishing_1 model Boundary Settings, Moving Mesh (ALE): boundaries 3, 4, 6, 7 win-

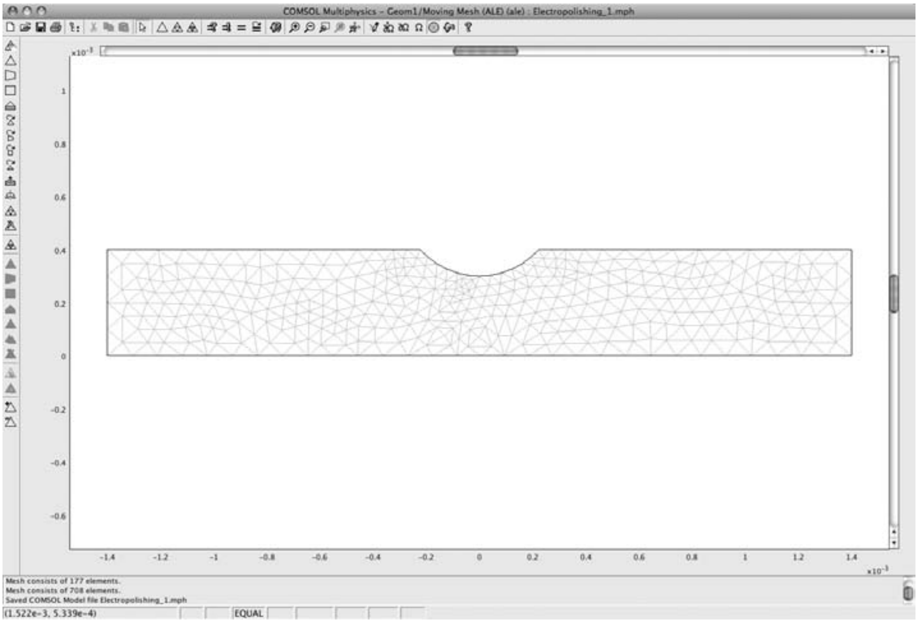


FIGURE 4.16 2D Electropolishing_1 model mesh

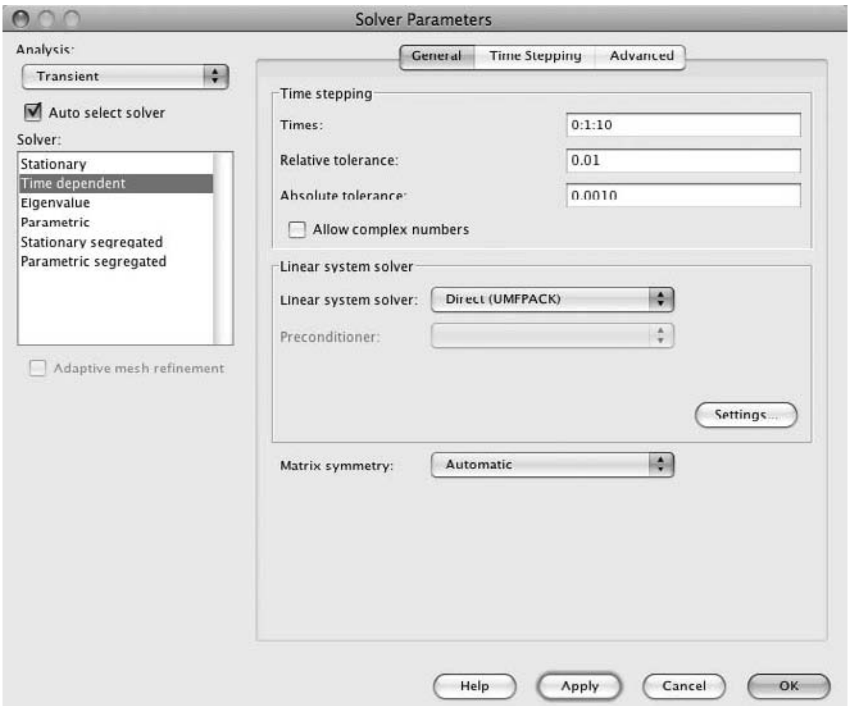


FIGURE 4.17 2D Electropolishing_1 model Solver Parameters window

---

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, always change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can try a variation on the model parameter space to seek improved results.

Using the menu bar, select Solve > Solve Problem.

Postprocessing

Select Postprocessing > Plot Parameters. Click the Surface tab, and verify that the Surface plot check box is checked.

From the Predefined quantities drop-down list, select "Conductive Media DC (dc) > Total current density, norm." See Figure 4.18.

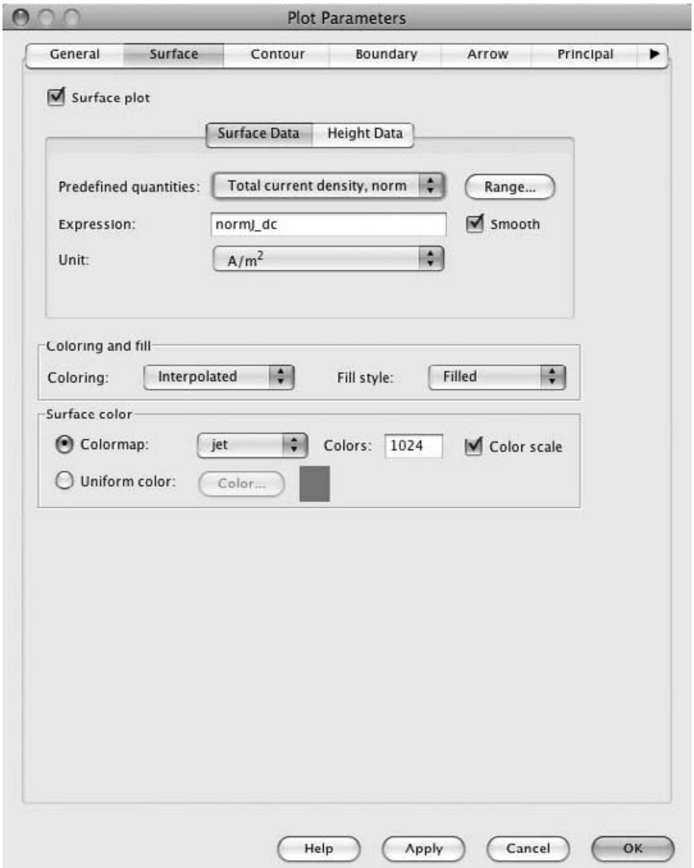


FIGURE 4.18 2D Electropolishing_1 model Plot Parameters window, Surface page

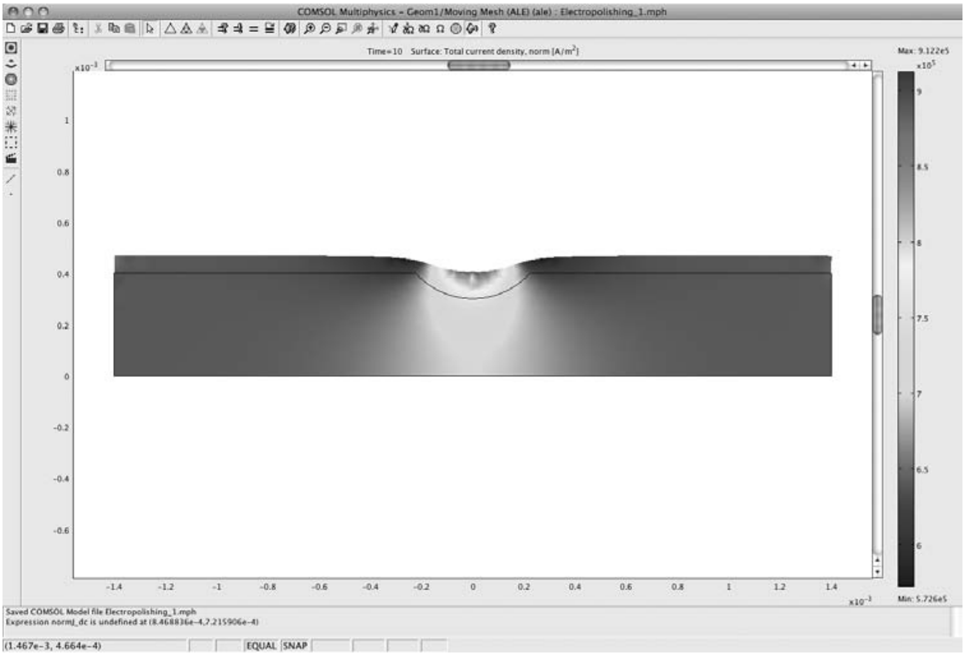


FIGURE 4.19 2D Electropolishing_1 model Surface plot window

Click OK. See Figure 4.19.

In Figure 4.19, the model calculation shows that the maximum current density is approximately 0.92e5 A/m^2, in the region of the asperity. Figure 4.19 also shows that the normal current density (J_n) concentrated in the region of the asperity is approximately 1.5 times the normal current density elsewhere on the electrode surface. As a result, the removal rate of the electrode material will be approximately 1.5 times as high.

To see the change in the position of the electrode surface and the relative removal of material from the asperity, first select Postprocessing > Plot Parameters. Next, click the Surface tab, and verify that the Surface plot check box is checked.

From the Predefined quantities drop-down list, select "Moving Mesh (ALE) (ale) > y-displacement." See Figure 4.20.

Click OK. Figure 4.21 shows the displacement of the electrode surface in the *y*-direction (dy_ale) after 10 seconds of electropolishing.

In Figure 4.21, the model calculation shows that the maximum displacement of the electrode surface in the *y*-direction (dy_ale) after 10 seconds of electropolishing is approximately 1.08e-4 m (0.108 mm), in the region of the asperity.

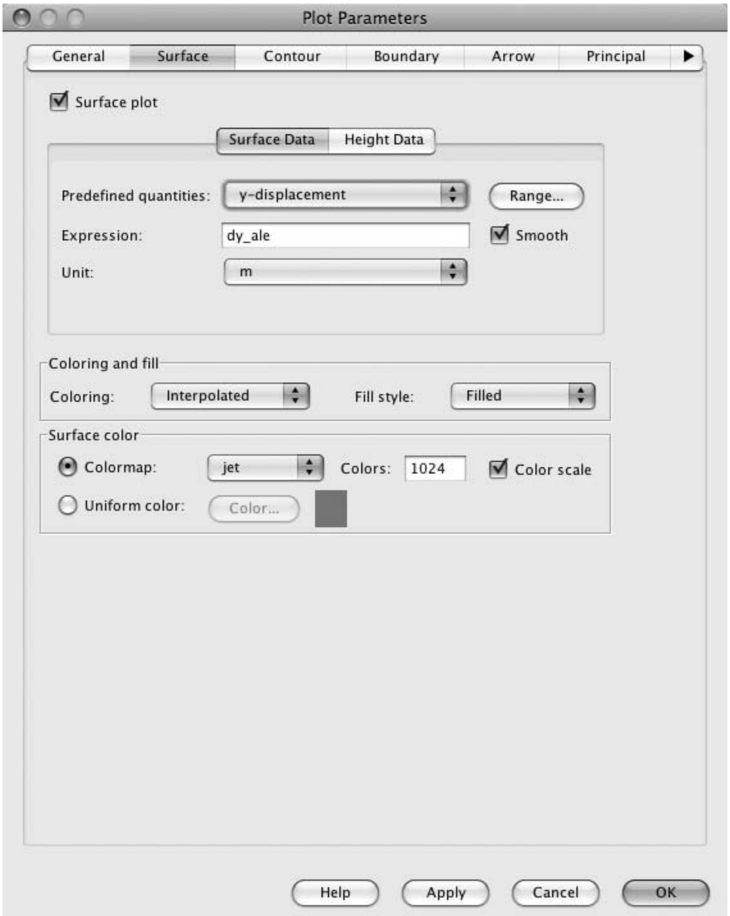


FIGURE 4.20 2D Electropolishing_1 model Surface plot window: Moving Mesh (ALE) (ale), y displacement (dy_ale)

NOTE The result of the modeling calculation is approximately 1.1e-4 m. Calculating the estimated result on a "first principles" basis:

$$d = |U|\Delta t = K|J_n|\Delta t = \left(10^{-11}\frac{\mathrm{m}^3}{\mathrm{A*s}}\right)*\left(10^6\frac{\mathrm{A}}{\mathrm{m}^2}\right)*(10^1\mathrm{s}) = 10^{-4}\mathrm{m} \qquad (4.3)$$

This agrees well with the results of the model.

Postprocessing Animation

This solution to the 2D electrochemical polishing model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing >

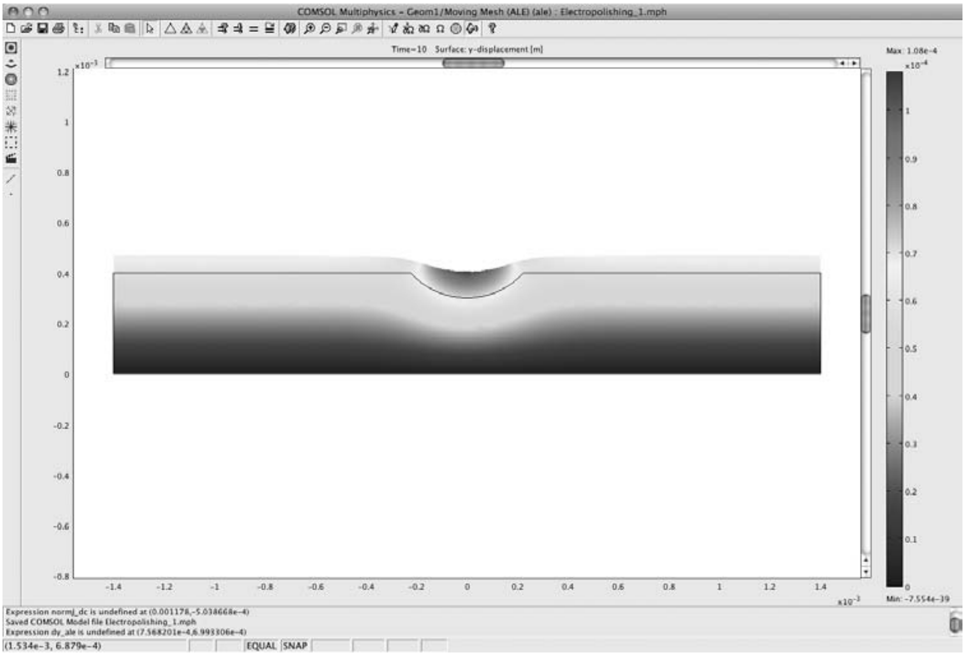


FIGURE 4.21 2D Electropolishing_1 model Surface plot window: Moving Mesh (ALE) (ale), y displacement (dy_ale)

Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 4.22). Click the Start Animation button. Save this 2D electrochemical polishing model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4_EP_1.avi that was supplied with this book.

First Variation on the 2D Electrochemical Polishing Model

NOTE This model will explore the effect of the mesh element type (triangle, quadrilateral [quad]) on the ultimate values determined by the calculated solution. Both the model geometry and the model mesh play major roles in the ease of solving any particular problem.

To start building the Electropolishing_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select “2D” from the Space dimension pull-down list. Select COMSOL Multiphysics > Deformed Mesh > Moving Mesh (ALE) > Transient analysis. Click the Multiphysics button, and then click the Add button.

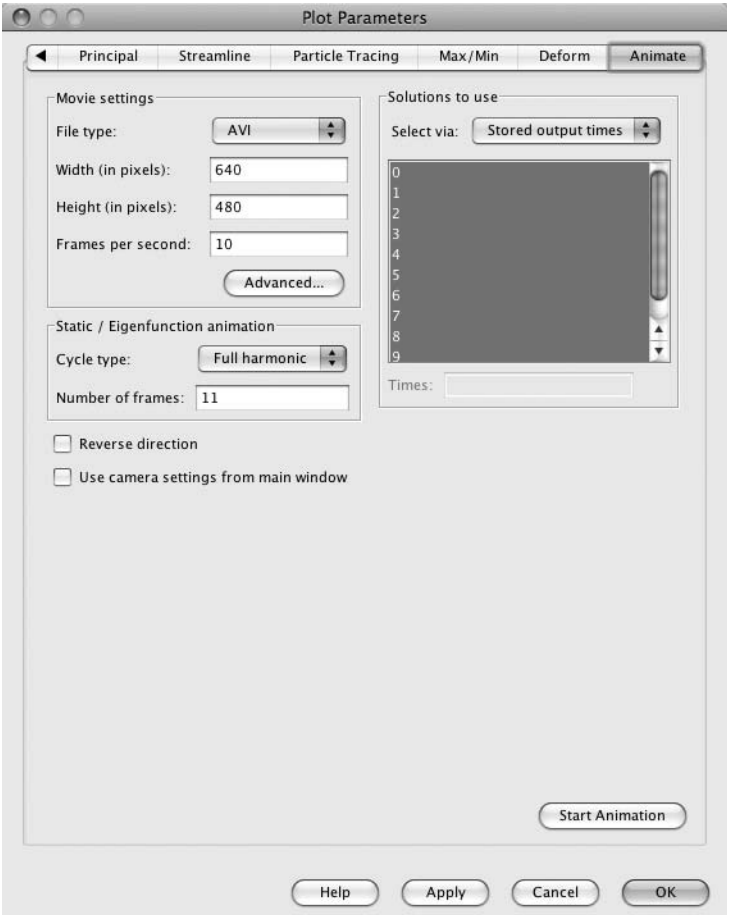


FIGURE 4.22 2D Electropolishing_1 model animation Plot Parameters window

Using the Application Modes list, select COMSOL Multiphysics > Electromagnetics > Conductive Media DC. Click the Add button. See Figure 4.23.

Click OK. Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.4; see Figure 4.24.

Table 4.4 Constants Edit Window

Name	Expression	Description
K	1.0e-11[m^3/(A*s)]	Coefficient of proportionality

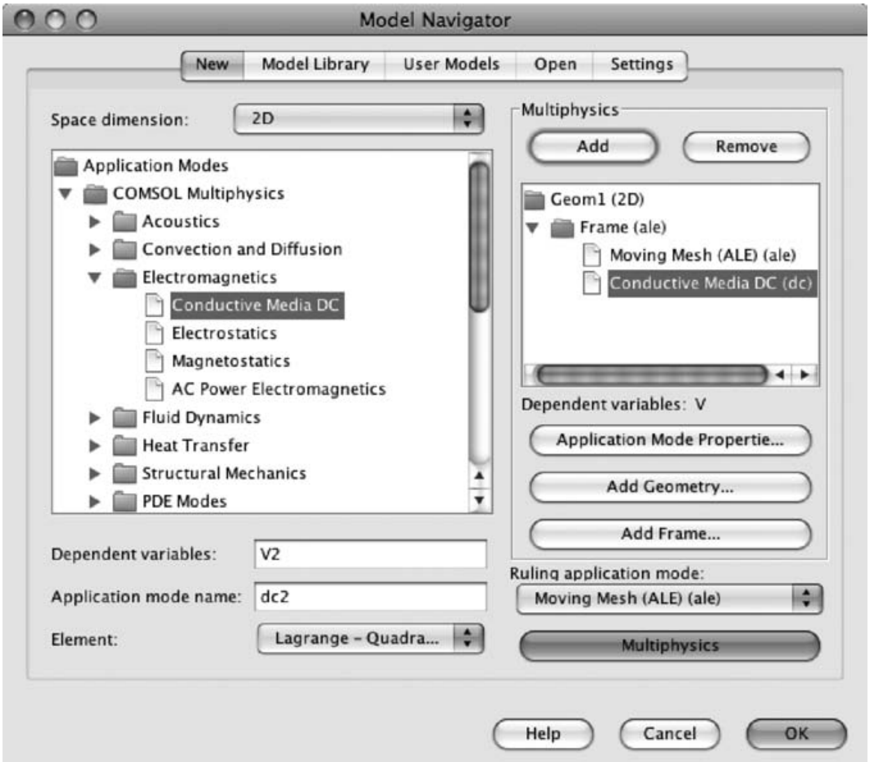


FIGURE 4.23 2D Electropolishing_2 Model Navigator setup

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 2.8 and a height of 0.4. Select "Base: Corner" and set X equal to −1.4 and Y equal to 0 in the Rectangle edit window. See Figure 4.25.

Click the Apply button, and then click OK. See Figure 4.26.

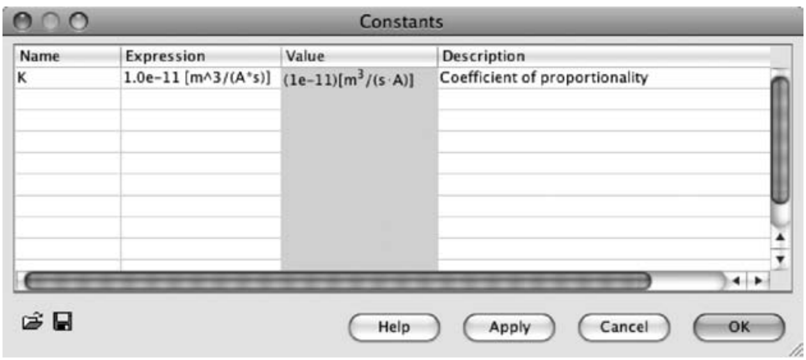


FIGURE 4.24 2D Electropolishing_2 model Constants edit window

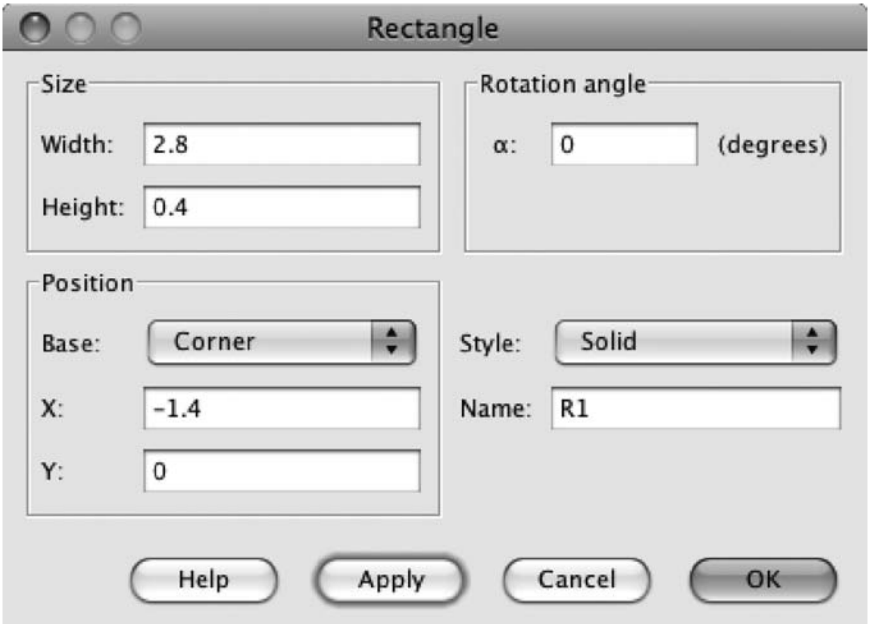


FIGURE 4.25 2D Electropolishing_2 model Rectangle edit window

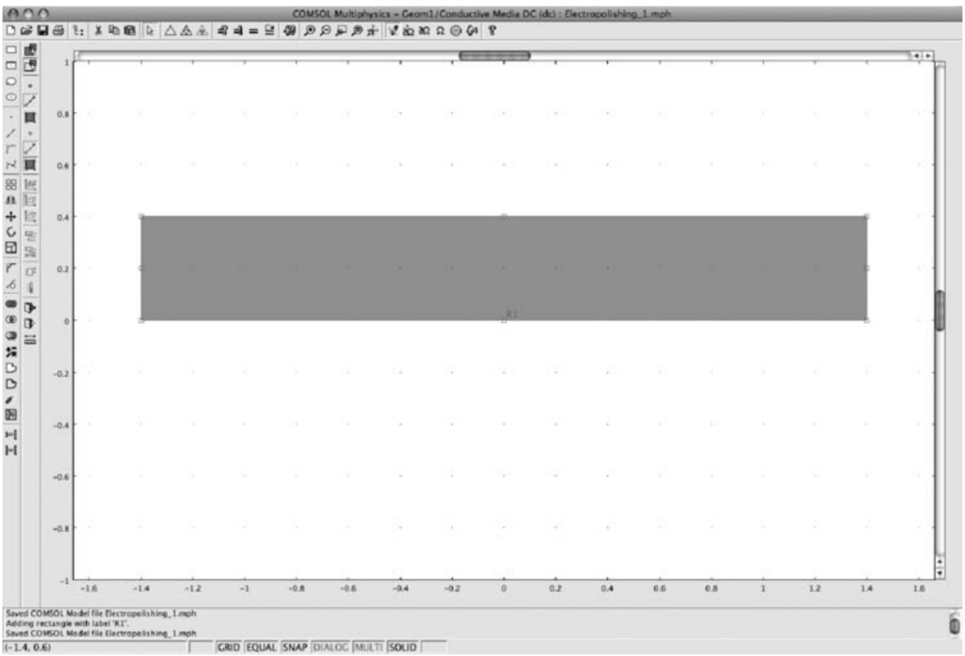


FIGURE 4.26 2D Electropolishing_2 model electrolyte rectangle

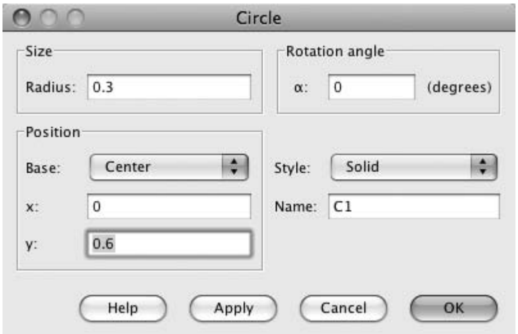


FIGURE 4.27 2D Electropolishing_12 model Circle edit window

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.3. select "Base: Center" and set X equal to 0 and Y equal to 0.6 in the Circle edit window. See Figure 4.27.

Click OK. See Figure 4.28.

Select both the rectangle and the circle by clicking on the rectangle and Shift-clicking on the circle. See Figure 4.29.

Click the Difference button on the Draw toolbar to remove the overlapping portion of the circle from the rectangle. The upper surface of the electrolyte rectangle (CO1) is the lower surface of the electrode, with the asperity, that will be electropolished. See Figure 4.30.

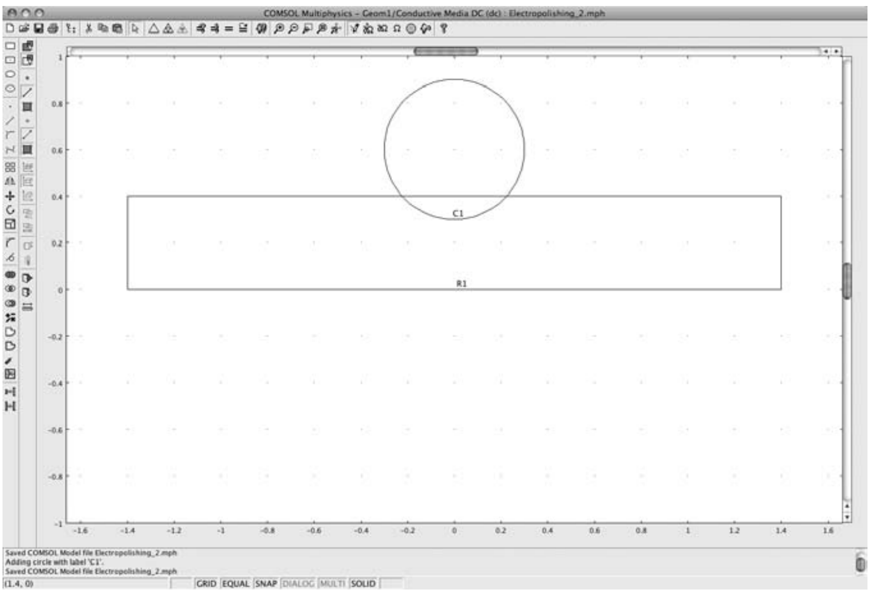


FIGURE 4.28 2D Electropolishing_12 model rectangle and circle

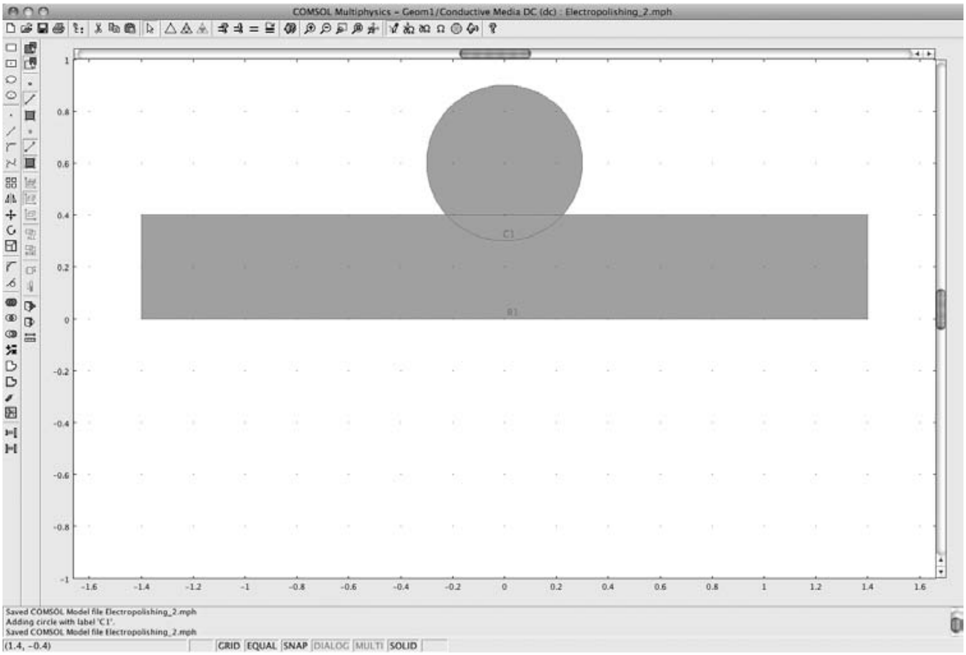


FIGURE 4.29 2D Electropolishing_12 model selected rectangle and circle

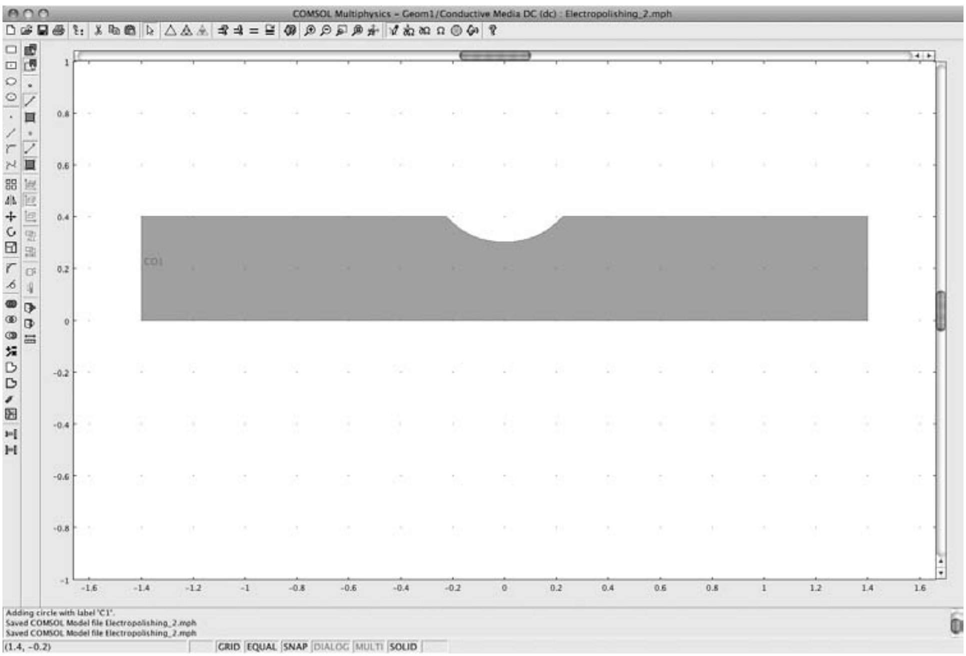


FIGURE 4.30 2D Electropolishing_12 model electrode with asperity

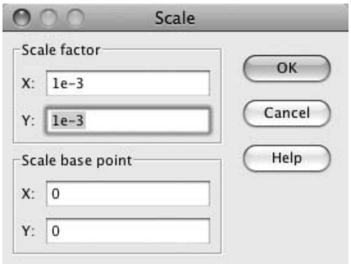


FIGURE 4.31 2D Electropolishing_12 model Scale edit window

NOTE The model geometry, as presently scaled, is 1.4 meters in length and 0.4 meter in height. Electropolishing is typically applied as the final finishing (smoothing) step in a precision fabrication process (e.g., metallographic samples, vacuum chambers). Thus the model geometry will need to be reduced in scale to emulate reality.

Click on the text "CO1." Click the Scale button on the Draw toolbar. Enter 1e-3 in both the X and Y text boxes in the Scale factor edit windows. See Figure 4.31.

Click OK, and then click the Zoom Extents button on the menu bar. See Figure 4.32.

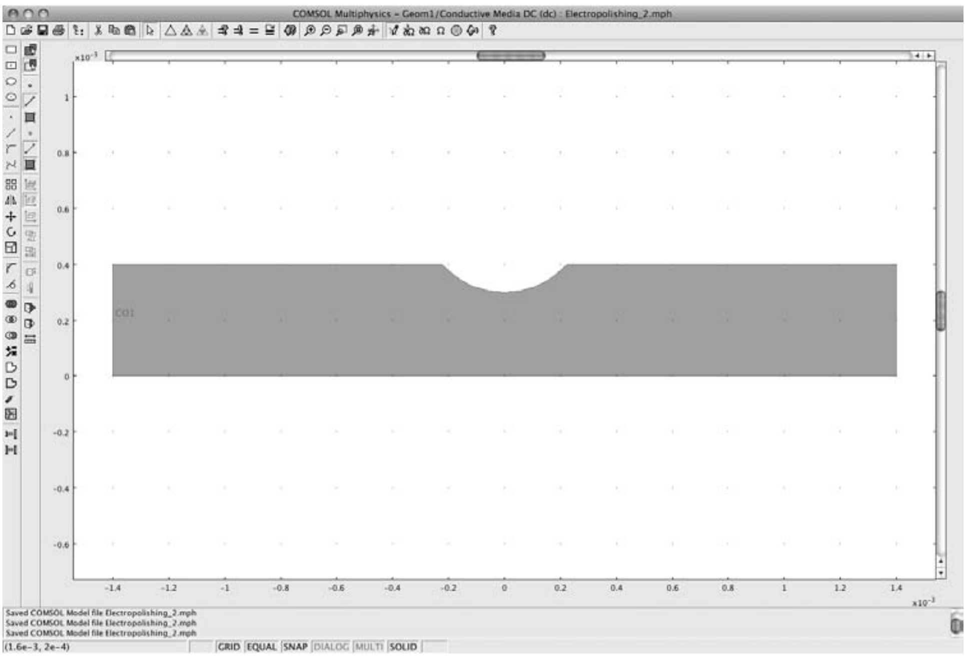


FIGURE 4.32 2D Electropolishing_12 model scaled electrolyte/electrode geometry

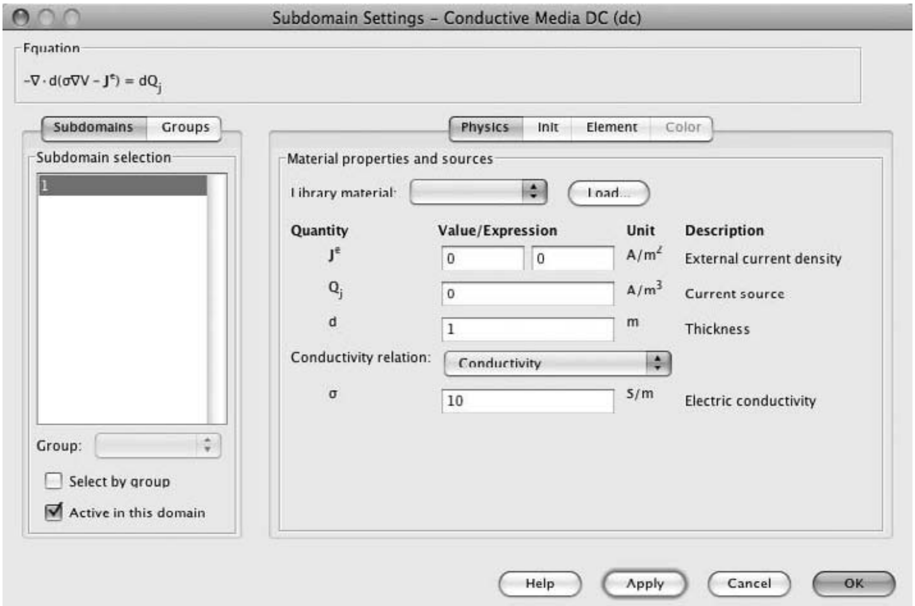


FIGURE 4.33 Subdomain Settings window

Physics Subdomain Settings: Conductive Media DC

Having established the 2D geometry for the electrochemical polishing model (a rectangle with a negative asperity on the upper surface), the next step is to define the fundamental physics conditions. Using the menu bar, select Multiphysics > Conductive Media DC. Next, using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). Enter 10 in the Electric conductivity (σ) edit window. See Figure 4.33. Click OK.

Physics Boundary Settings: Conductive Media DC

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value as shown in Table 4.5. See Figures 4.34, 4.35, and 4.36.

Table 4.5 Boundary Settings, Conductive Media DC Window

Boundary	Boundary Condition	Value/Expression
1, 5	Electric insulation	—
3, 4, 6, 7	Electric potential	30
2	Ground	—

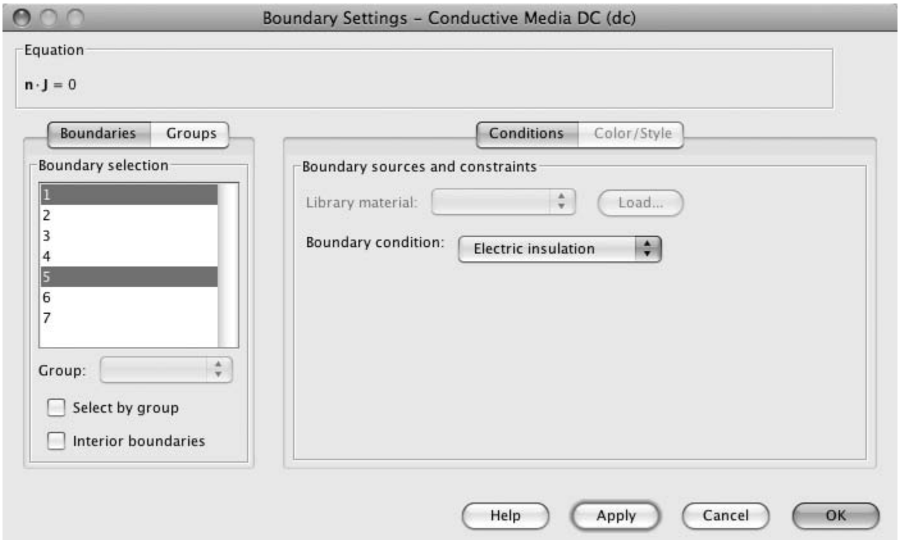


FIGURE 4.34 Boundary Settings (1, 5), Conductive Media DC: boundaries set

Click OK. See Figure 4.37.

Physics Boundary Settings: Moving Mesh (ALE)

Using the menu bar, select Multiphysics > Moving Mesh (ALE). Next, using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or

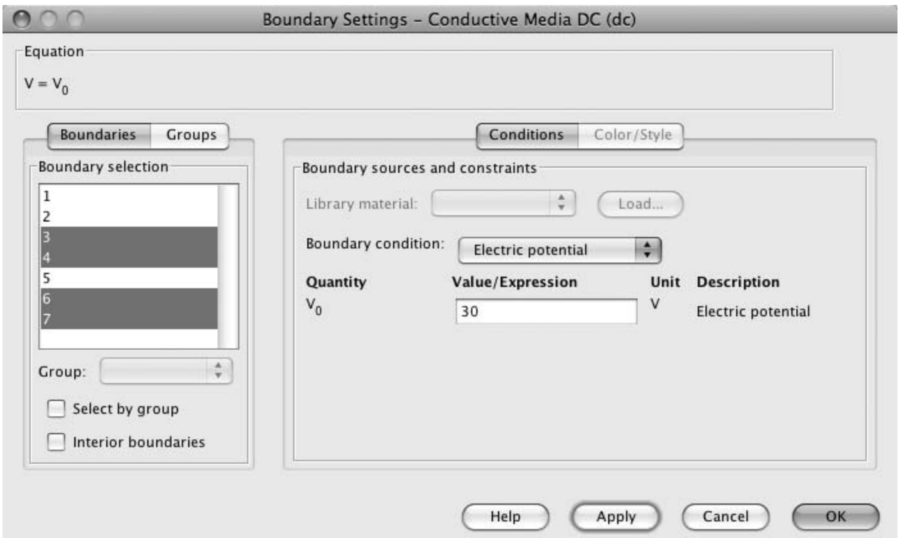


FIGURE 4.35 Boundary Settings (3, 4, 6, 7), Conductive Media DC: boundaries set

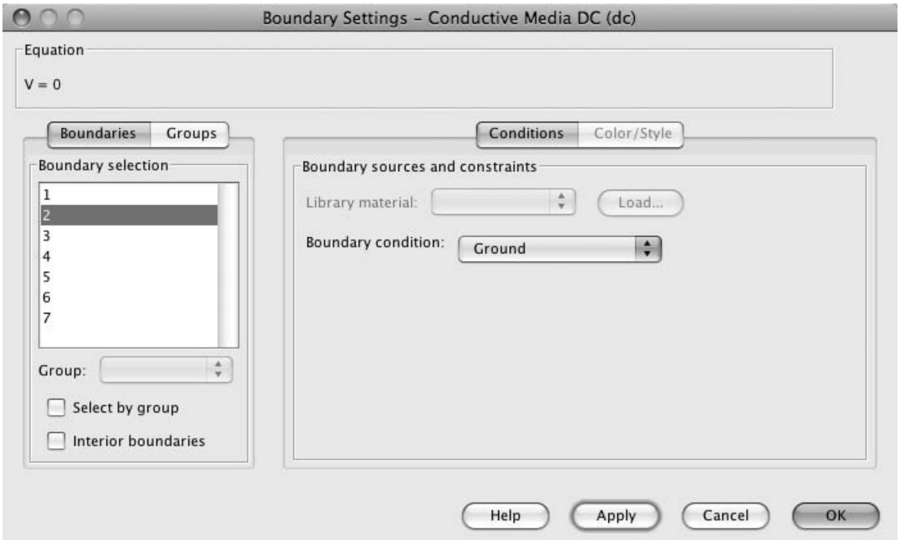


FIGURE 4.36 Boundary Settings (2), Conductive Media DC: boundaries set

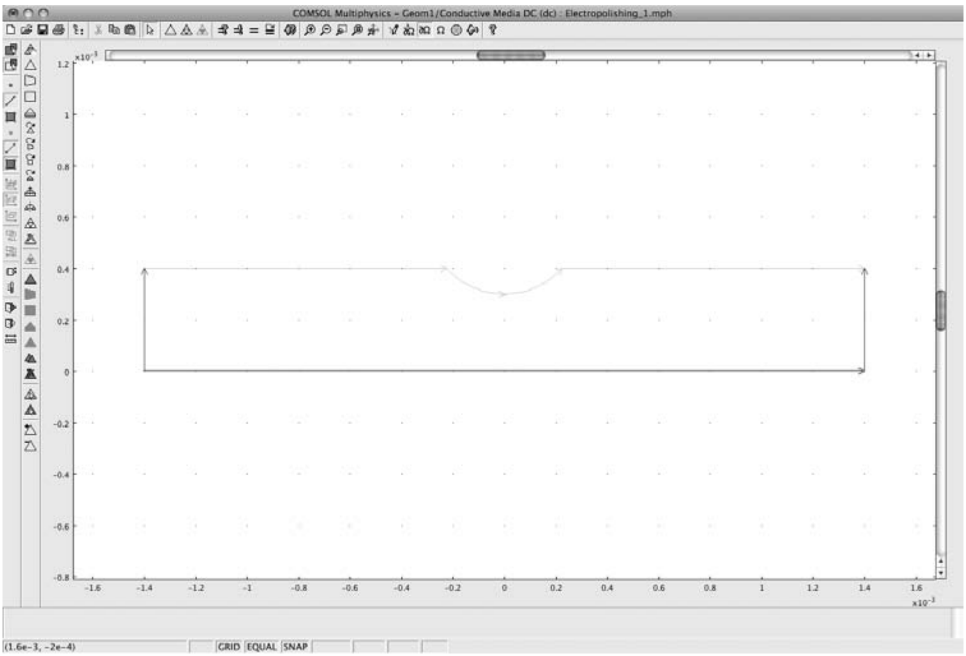


FIGURE 4.37 Boundary Settings (1, 5 = blue; 3, 4, 6, 7 = green; 2 = red), Conductive Media DC: boundaries set

Table 4.6 Boundary Settings, Moving Mesh (ALE) Window

Boundary	Coordinate	Boundary Condition	Value/Expression
1, 5	Global	Mesh velocity	vx = 0
3, 4, 6, 7	Tangent and normal Deformed mesh	Mesh velocity	vn = −K*nJ_dc
2	Global	Mesh displacement	dx = 0, dy = 0

enter the given boundary condition and value in the edit windows as indicated in Table 4.6. See Figure 4.38, 4.39, and 4.40.

Click OK. Figure 4.41 shows the Boundary Settings, Moving Mesh (ALE) options organized by color.

Mesh Generation

From the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 1 in the Subdomain selection window. Enter 4e-5 in the Maximum element size edit window. Select "Quad" from the Method drop-down list. See Figure 4.42.

NOTE The model default mesh, in the COMSOL Multiphysics software, is the triangular mesh. The triangular mesh is simpler and generates fewer parameters to calculate. However, the quad mesh may be a better mathematical fit to the model for which a solution is sought. The modeler needs to decide the most appropriate choice for the model under consideration.

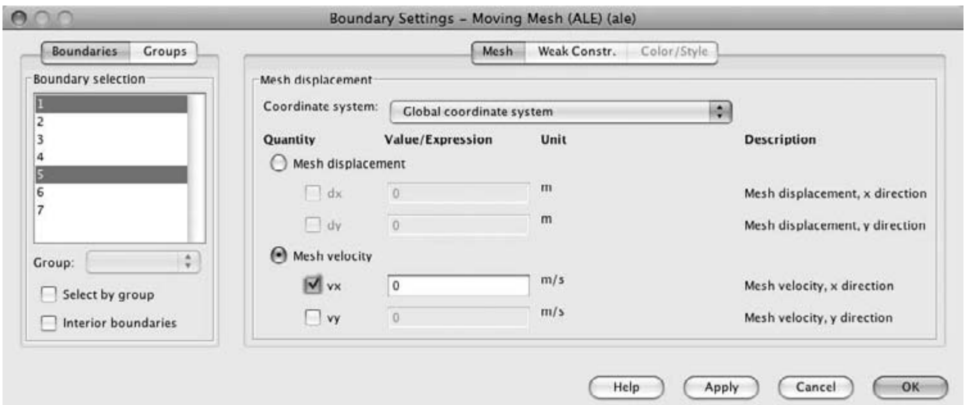


FIGURE 4.38 Boundary Settings (1, 5), Moving Mesh (ALE): Boundaries window

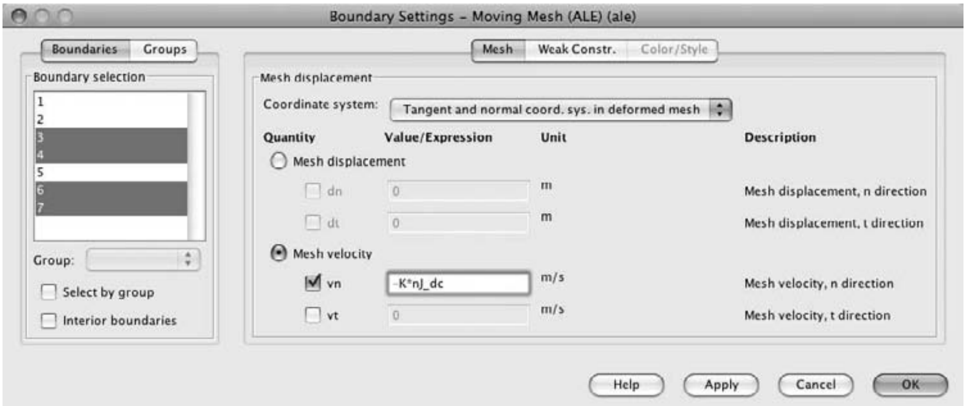


FIGURE 4.39 Boundary Settings (3, 4, 6, 7), Moving Mesh (ALE): Boundaries window

For a simple 2D model, the maximum element size value for a particular subdivision can be estimated by dividing the lesser ($A < B$) dimension by 10 and then testing how the calculated model satisfies the goals of the modeler.

Click the Remesh button, and then click OK. See Figure 4.43. This mesh contains approximately 754 elements.

Solving the First Variation on the 2D Electrochemical Polishing Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver. Enter 0:0.5:10 in the Times edit window, as shown in Figure 4.44. This instruction causes the Solver to divide the modeling time-space into 20 equal intervals, over the period from 0 to 10 seconds. Click OK.

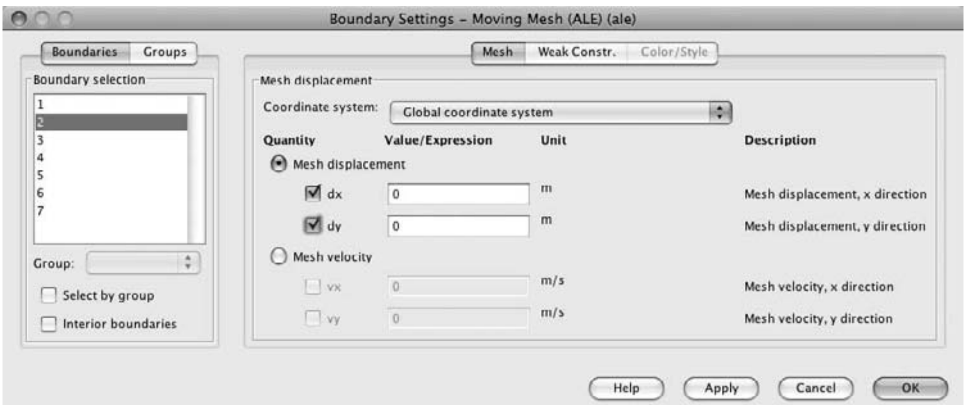


FIGURE 4.40 Boundary Settings (2), Moving Mesh (ALE): Boundaries window

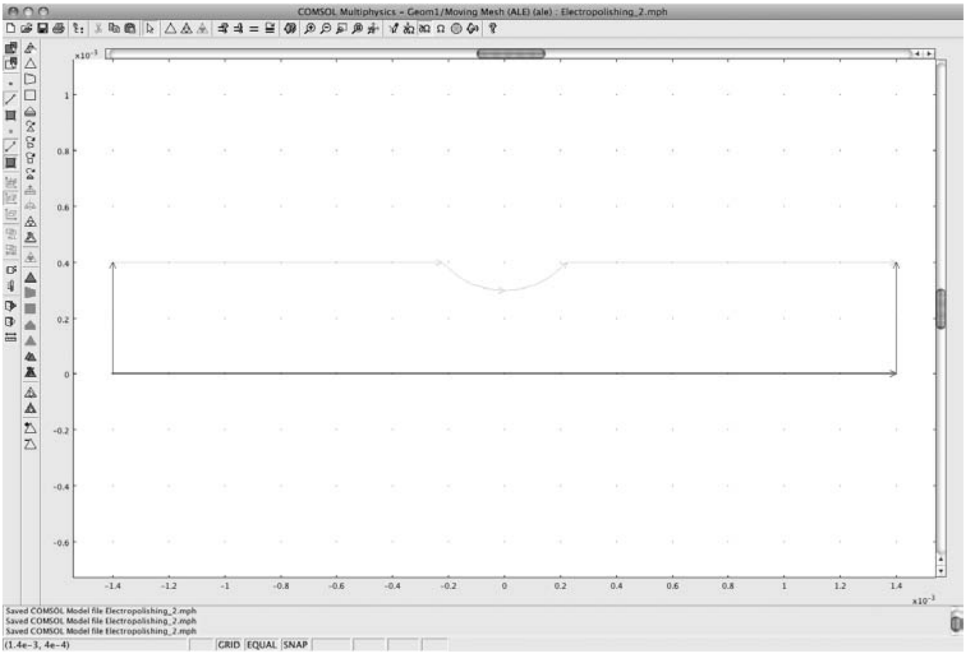


FIGURE 4.41 Boundary Settings, Moving Mesh (ALE): boundaries organized by color [green = tangent and normal coordinate system in deformed mesh; blue = global (vx = 0); red = global (dx = 0, dy = 0)]

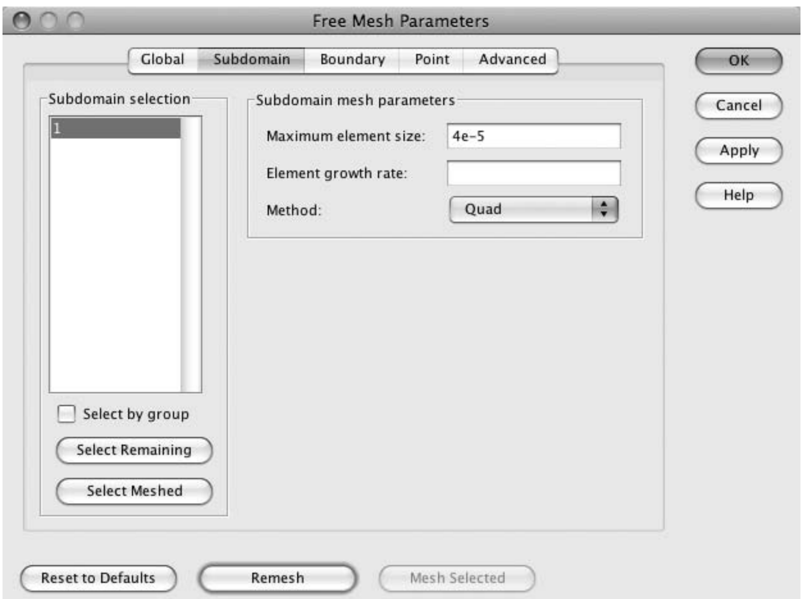


FIGURE 4.42 2D electrochemical polishing model Free Mesh Parameters window

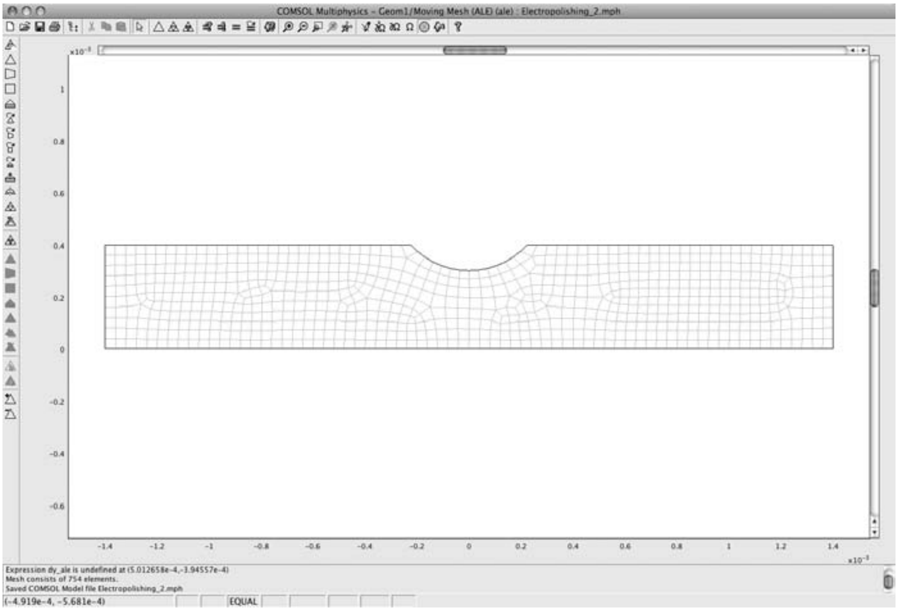


FIGURE 4.43 2D electrochemical polishing model free mesh (quad)

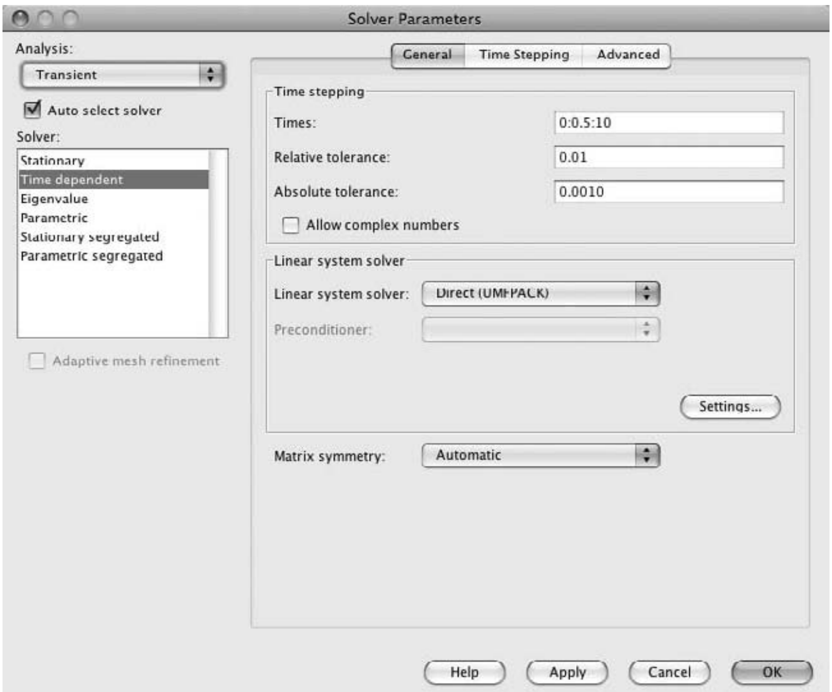


FIGURE 4.44 2D electrochemical polishing model Solver Parameters window

Using the menu bar, select Solve > Solve Problem.

NOTE In the process of solving this model, using the Moving Mesh Application Mode (ALE), the modeler may occasionally see a warning about an "inverted mesh element." If the solver continues on to the solution, ignore the warning. Such warnings are normal when using the deformed mesh.

If the model does not continue to a solution and the solver displays numerous warnings, then either there is an error in the model or the modeler needs to use the advanced technique called remesh (not discussed in this book).

Postprocessing

Select Postprocessing > Plot Parameters. Click the Surface tab, and verify that the Surface plot check box is checked. From the Predefined quantities drop-down list, select Conductive Media DC (dc) > Total current density, norm. See Figure 4.45. Click OK.

FIGURE 4.45 2D electrochemical polishing model Plot Parameters window

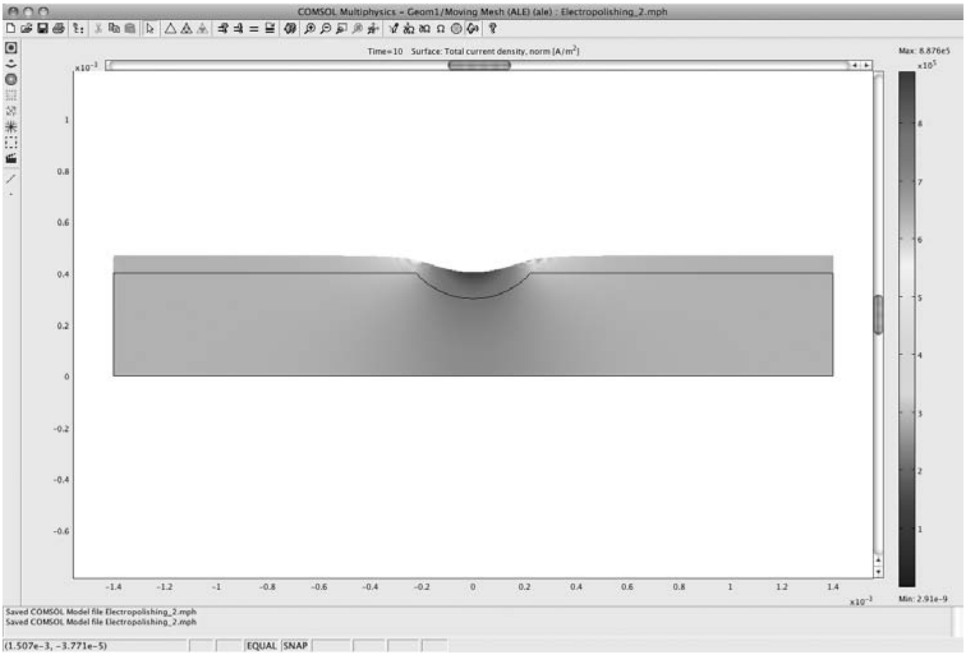


FIGURE 4.46 2D electrochemical polishing model Surface plot window, total normal current density

In Figure 4.46, the model calculation shows that the maximum current density is approximately 8.87e5 A/m^2, in the region of the asperity (very close to the calculated value in the original model). Figure 4.46 also shows that the normal current density (J_n) concentrated in the region of the asperity is approximately 1.5 times the normal current density elsewhere on the electrode surface. As a result, the removal rate of the electrode material will be approximately 1.5 times as high.

To see the change in the position of the electrode surface and the relative removal of material from the asperity, select Postprocessing > Plot Parameters. Click the Surface tab, and verify that the Surface plot check box is checked. From the Predefined quantities drop-down list, select Moving Mesh (ALE) (ale) > y-displacement. See Figure 4.47.

Click OK. Figure 4.48 shows the displacement of the electrode surface in the *y*-direction (dy_ale) after 10 seconds of electropolishing.

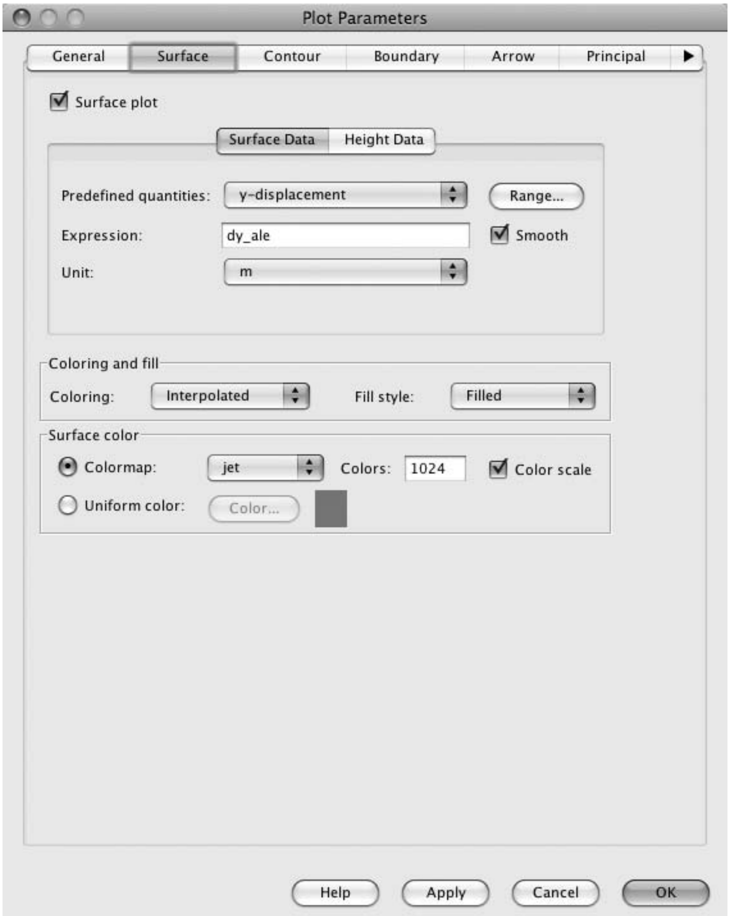


FIGURE 4.47 2D electrochemical polishing model surface Plot Parameters window: Moving Mesh (ALE) (ale), y direction (dy_ale)

Postprocessing Animation

This solution to the 2D electrochemical polishing model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 4.49). Click the Start Animation button. Save this 2D electrochemical polishing model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4_EP_2.avi that was supplied with this book.

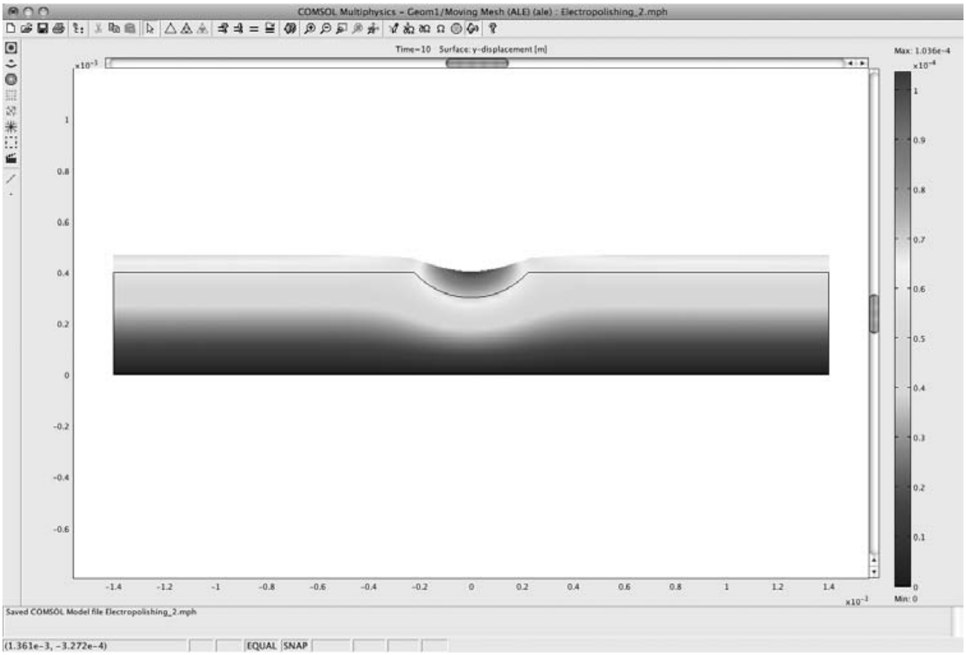


FIGURE 4.48 2D electrochemical polishing model Surface plot window: Moving Mesh (ALE) (ale), y direction (dy_ale)

Second Variation on the 2D Electrochemical Polishing Model

NOTE This model will explore the effect of a change in the shape and number of asperities. In this model, the quad mesh element type will be used, based on the excellent values calculated in the previous modeling solution. Bear in mind that both the model geometry and the model mesh play major roles in the ease of solving any particular problem.

To start building the Electropolishing_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select “2D” from the Space dimension pull-down list. Select COMSOL Multiphysics > Deformed Mesh > Moving Mesh (ALE) > Transient analysis. Click the Multiphysics button, and then click the Add button.

Using the Application Modes list, select COMSOL Multiphysics > Electromagnetics > Conductive Media DC. Click the Add button. See Figure 4.50.

Click OK. Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.7; also see Figure 4.51.

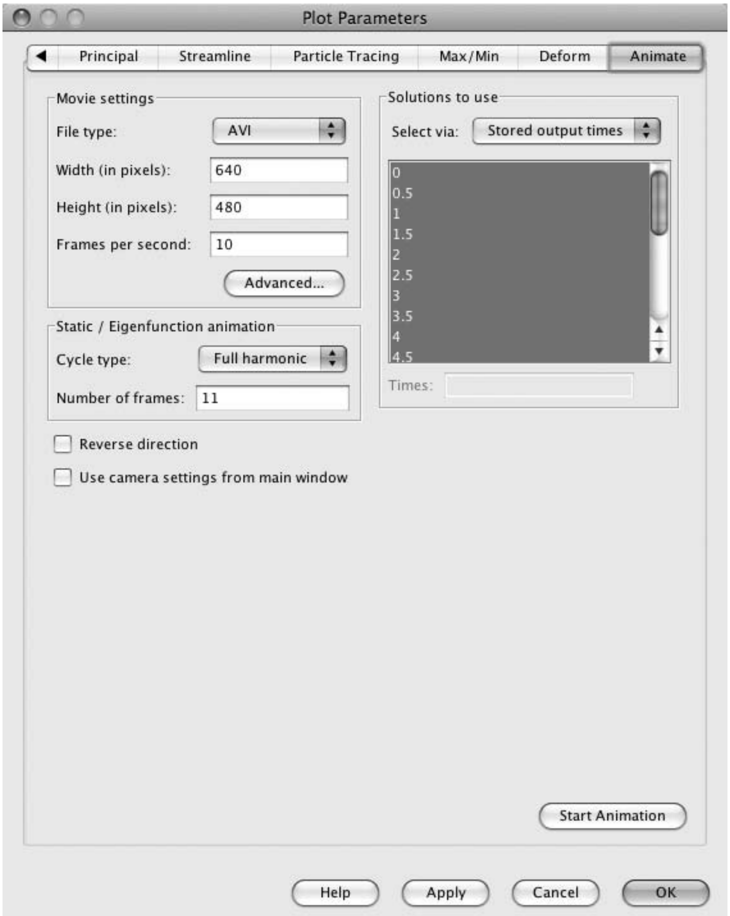


FIGURE 4.49 2D electrochemical polishing model animation Plot Parameters window

Table 4.7 Constants Edit Window

Name	Expression	Description
K	1.0e-11[m^3/(A*s)]	Coefficient of proportionality

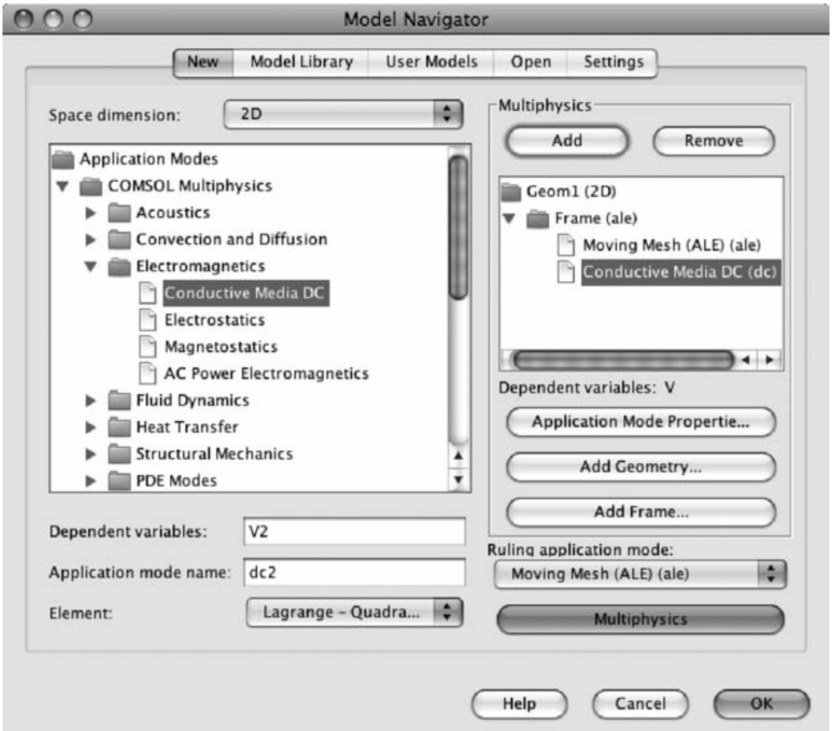


FIGURE 4.50 2D Electropolishing_3 Model Navigator setup

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 2.8 and a height of 0.4. Select "Base: Corner" and set X equal to −1.4 and Y equal to 0 in the Rectangle edit window. See Figure 4.52.

Click the Apply button, and then click OK. See Figure 4.53.

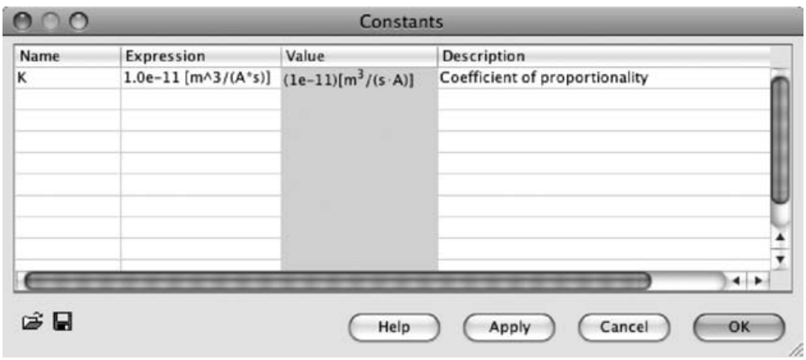


FIGURE 4.51 2D Electropolishing_3 model Constants edit window

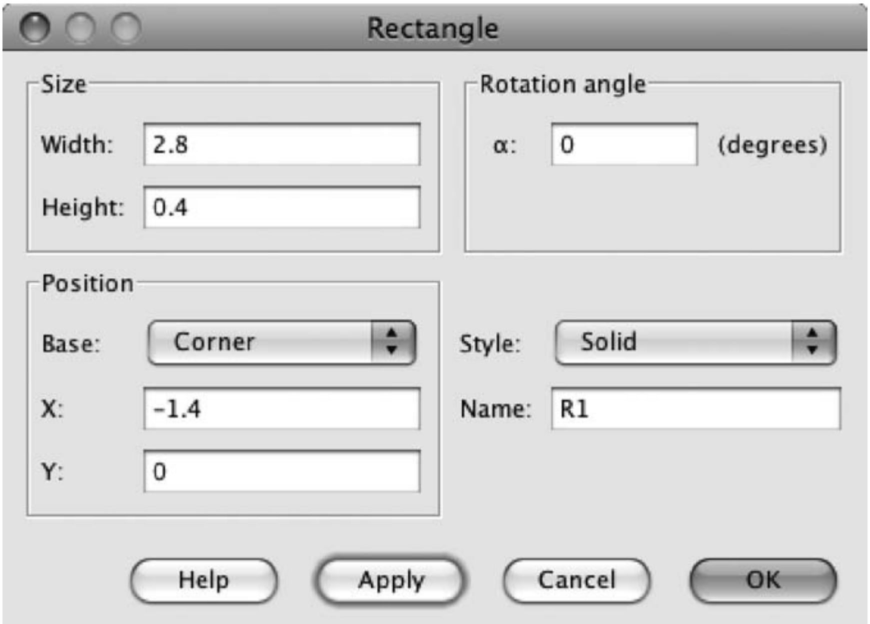


FIGURE 4.52 2D Electropolishing_3 model Rectangle edit window

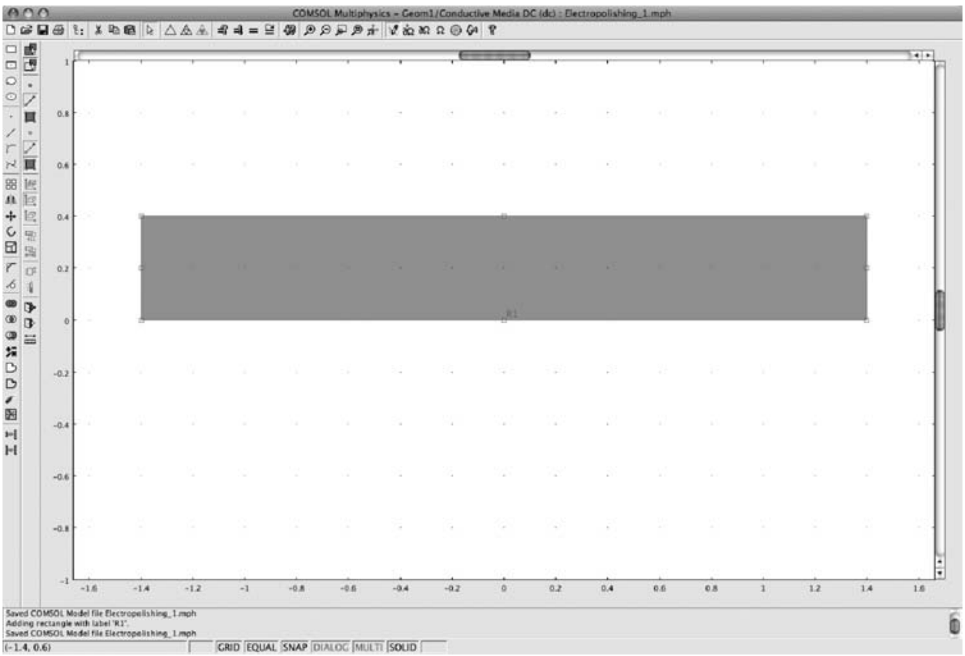


FIGURE 4.53 2D Electropolishing_3 model electrolyte rectangle

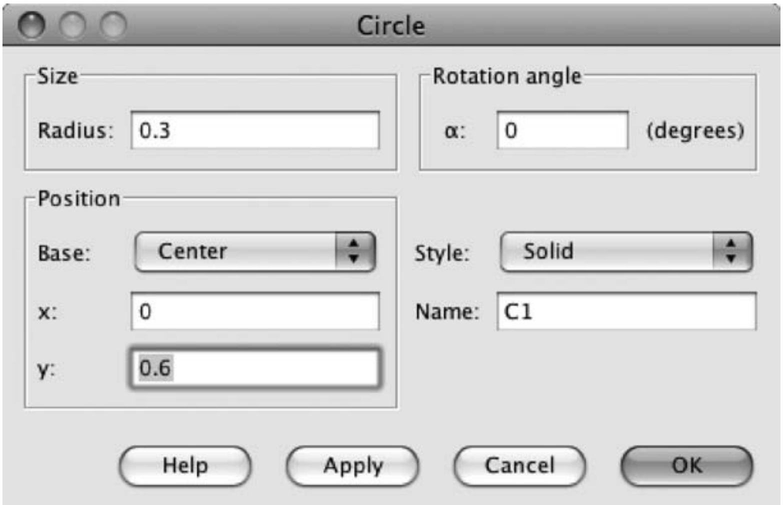


FIGURE 4.54 2D Electropolishing_3 model Circle edit window

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.3. Select "Base: Center" and set x equal to 0 and y equal to 0.6 in the Circle edit window. See Figure 4.54.

Click OK. See Figure 4.55.

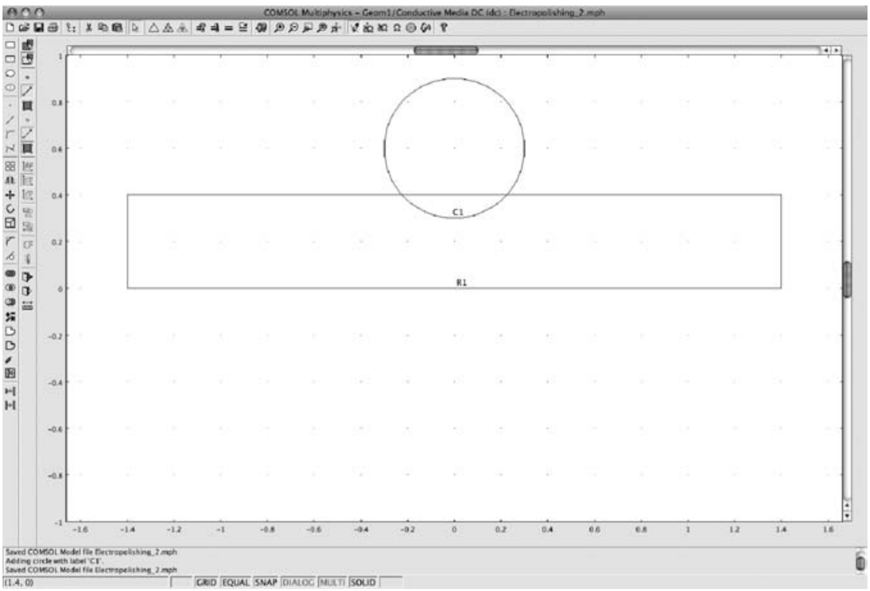


FIGURE 4.55 2D Electropolishing_3 model rectangle and circle

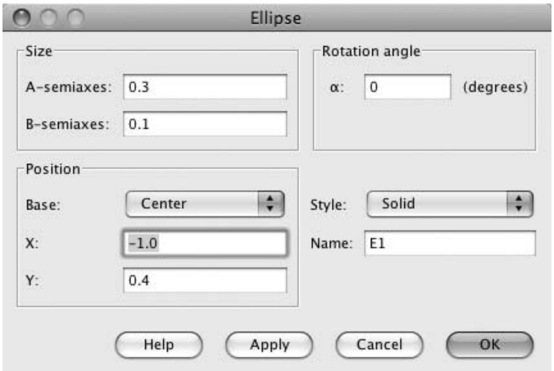


FIGURE 4.56 2D Electropolishing_3 model Ellipse edit window

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter 0.3 in the A-semiaxes edit window and 0.1 in the B-semiaxes edit window. Select “Base: Center” and set X equal to −1.0 and Y equal to 0.4 in the X and Y edit windows. See Figure 4.56.

Click OK. See Figure 4.57.

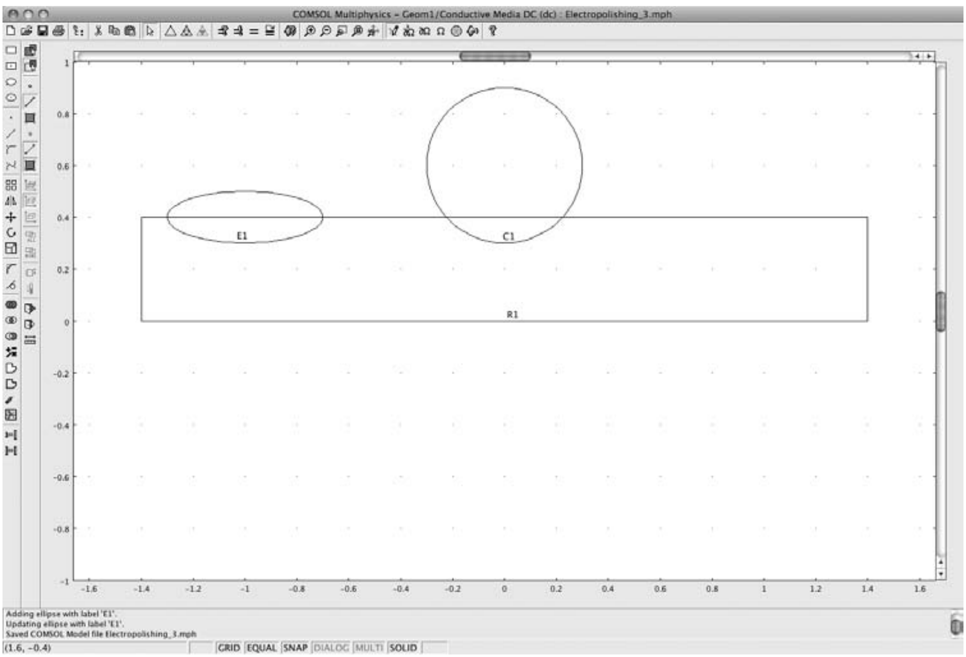


FIGURE 4.57 2D Electropolishing_3 model (C1, E1, R1)

FIGURE 4.58 2D Electropolishing_3 model Paste edit window

Select the text "E1." Using the menu bar, select Edit > Copy. Using the menu bar, select Edit > Paste. Enter 2.0 in the X: Displacements edit window. See Figure 4.58.

Click OK. See Figure 4.59.

Using the menu bar, select Draw > Create Composite Object. Enter R1-C1-E1-E2 in the Set formula edit window. See Figure 4.60.

NOTE To obtain the desired difference response, the modeler needs to key in the requested R1-C1-E1-E2 information in the edit window, rather than clicking on items in the Object selection window.

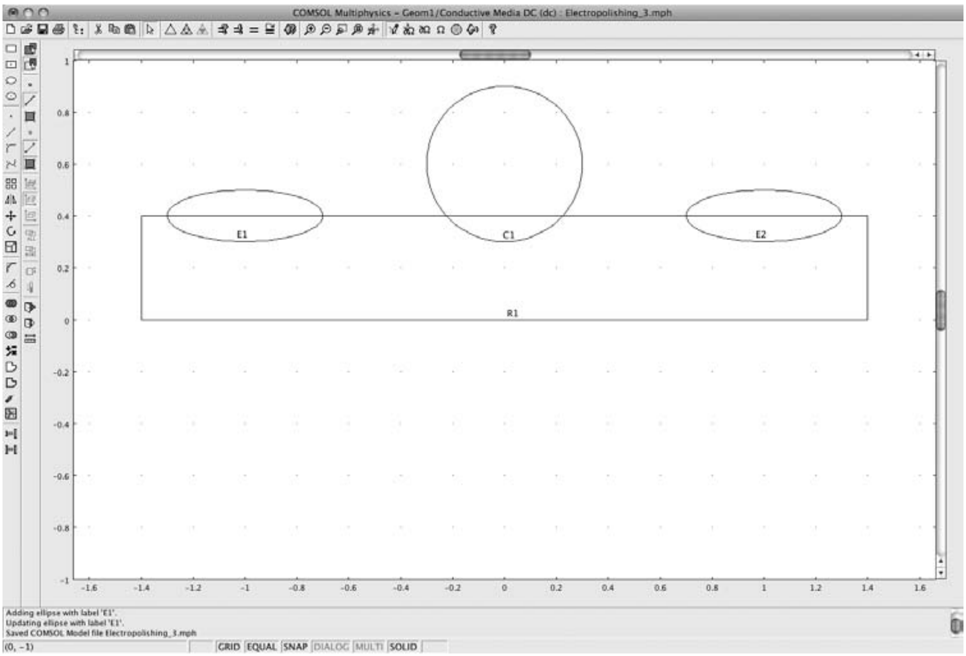


FIGURE 4.59 2D Electropolishing_3 model (C1, E1, E2, R1)

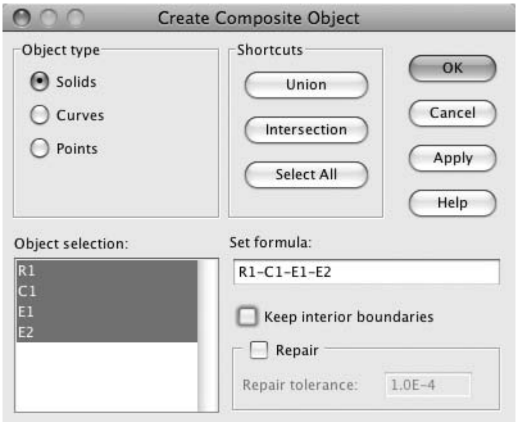


FIGURE 4.60 2D Electropolishing_3 model Create Composite Object edit window

Click OK. See Figure 4.61.

NOTE The model geometry, as presently scaled, is 1.4 meters in length and 0.4 meter in height. Electropolishing is typically applied as the final finishing (smoothing) step in a precision fabrication process (e.g., metallographic samples, vacuum chambers). Thus the model geometry will need to be reduced in scale to emulate reality.

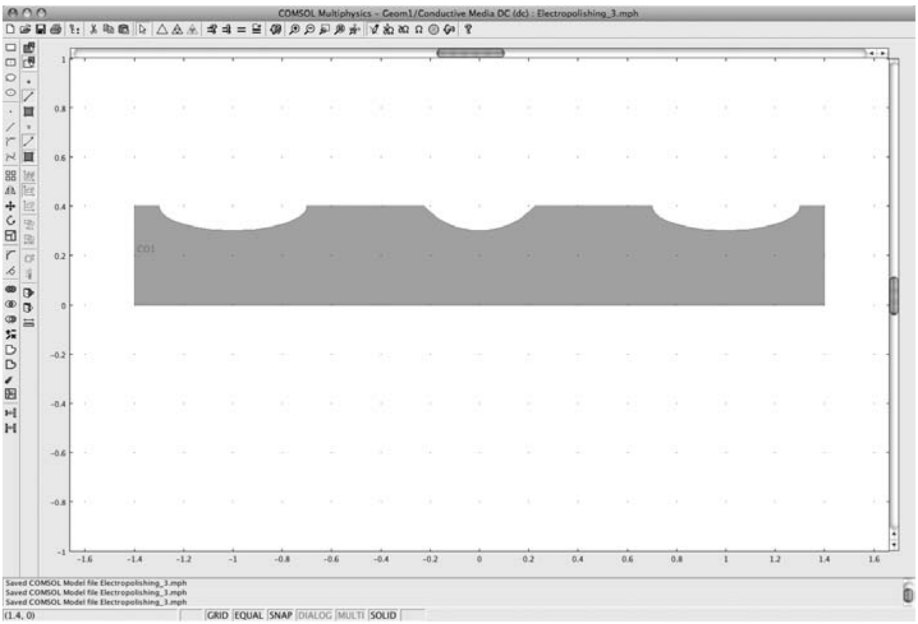


FIGURE 4.61 2D Electropolishing_3 model electrode with asperities

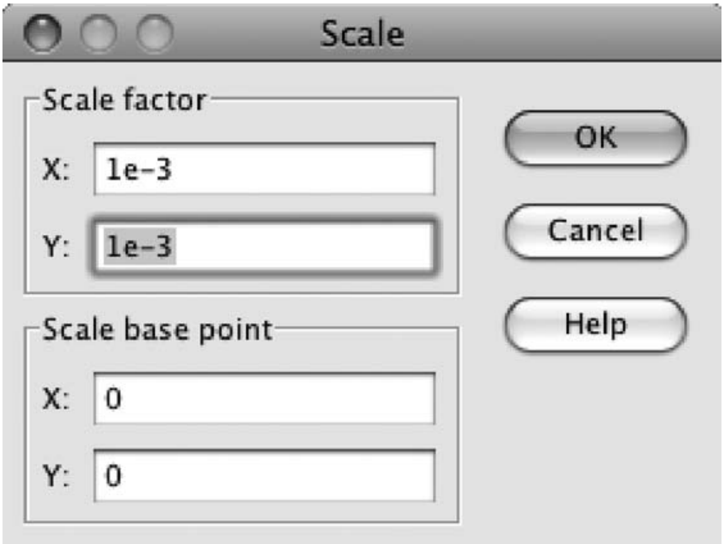


FIGURE 4.62 2D Electropolishing_3 model Scale edit window

Click on the text "CO1." Click the Scale button on the Draw toolbar. Enter 1e-3 in both the X and Y edit windows. See Figure 4.62.

Click OK, and then click the Zoom Extents button on the menu bar. See Figure 4.63.

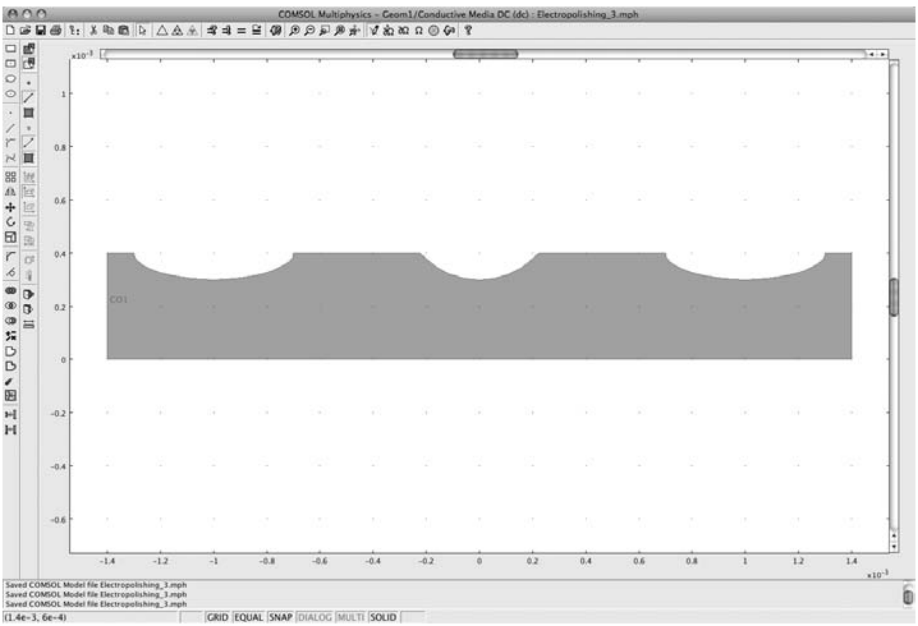


FIGURE 4.63 2D Electropolishing_3 model scaled electrolyte/electrode geometry

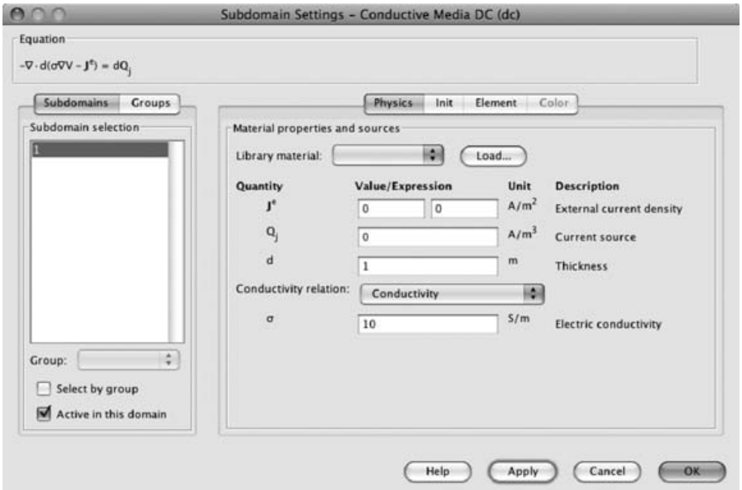


FIGURE 4.64 Subdomain Settings window

Physics Subdomain Settings: Conductive Media DC

Having established the 2D geometry for the electrochemical polishing model (a rectangle with negative asperities on the upper surface), the next step is to define the fundamental physics conditions. Using the menu bar, select Multiphysics > Conductive Media DC. Next, using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). Enter 10 in the Electric conductivity (σ) edit window. See Figure 4.64. Click OK.

Physics Boundary Settings: Conductive Media DC

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value as shown in Table 4.8. See Figures 4.65, 4.66, and 4.67.

Click OK. See Figure 4.68.

Table 4.8 Boundary Settings, Conductive Media DC Window

Boundary	Boundary Condition	Value/Expression
1, 7	Electric insulation	—
3–6, 8–13	Electric potential	30
2	Ground	—

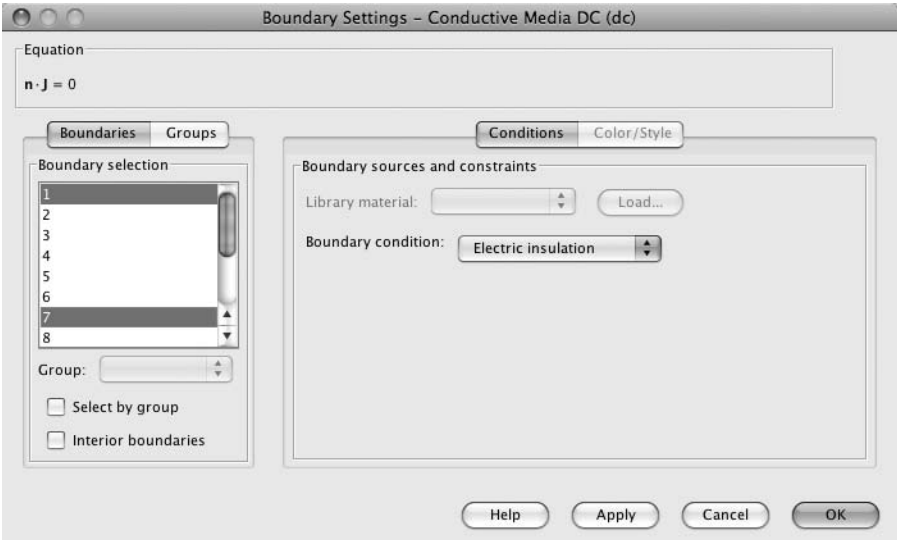


FIGURE 4.65 Boundary Settings (1, 7), Conductive Media DC: boundaries set

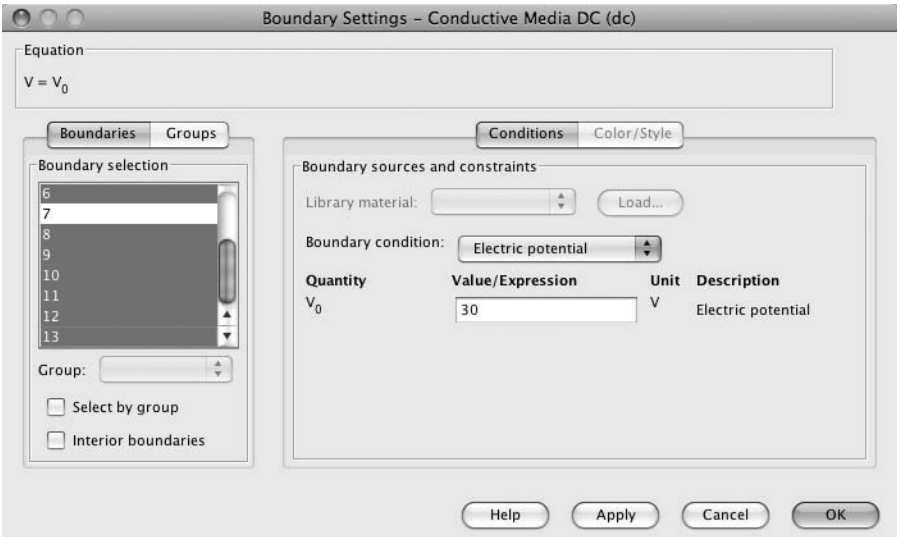


FIGURE 4.66 Boundary Settings (3–6, 8–13), Conductive Media DC: boundaries set

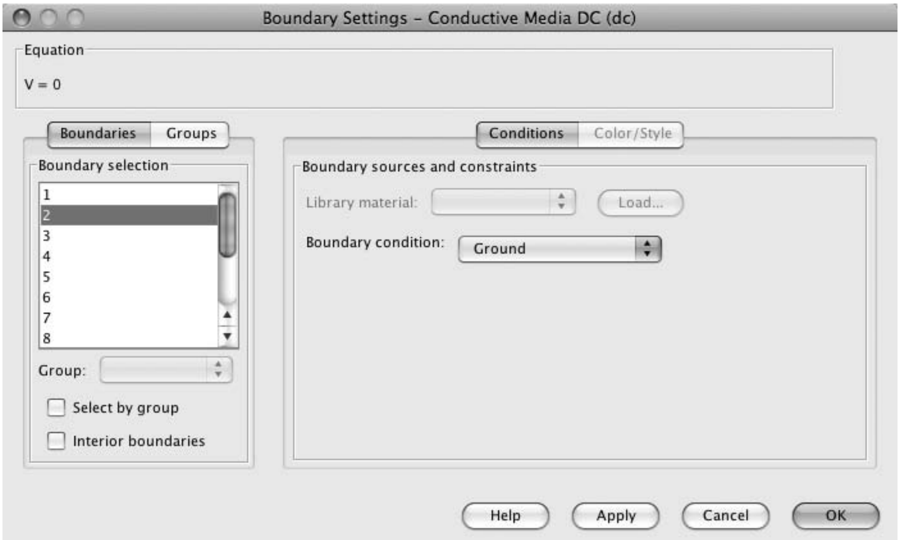


FIGURE 4.67 Boundary Settings (2), Conductive Media DC: boundary set

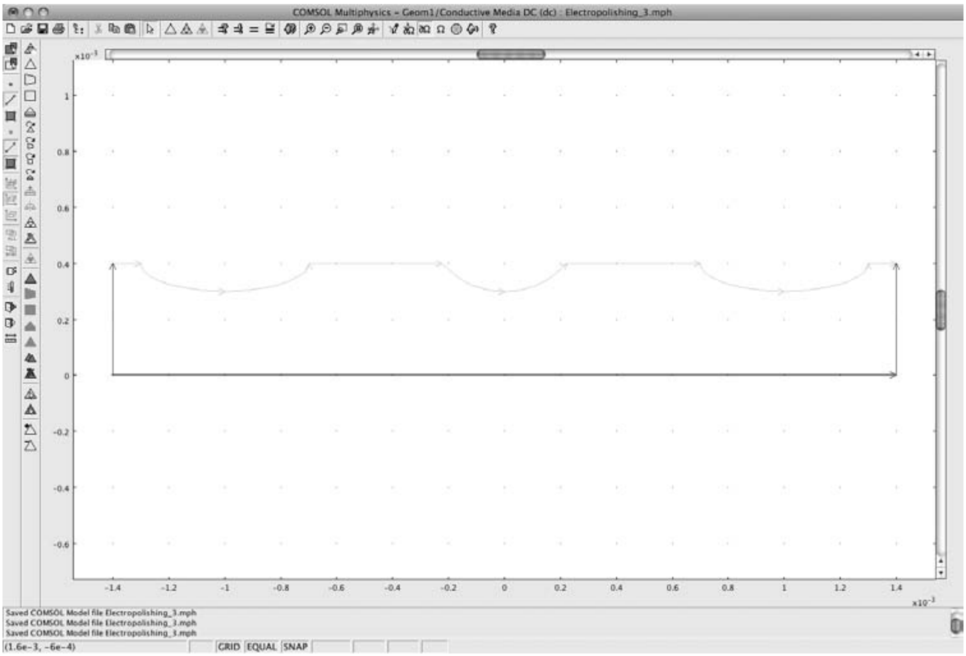


FIGURE 4.68 Boundary Settings (1, 7 = blue; 3–6, 8–13 = green; 2 = red), Conductive Media DC: boundaries set

Table 4.9 Boundary Settings, Moving Mesh (ALE) Window

Boundary	Coordinate	Boundary Condition	Value/Expression
1, 7	Global	Mesh velocity	vx = 0
3–6, 8–13	Tangent and normal Deformed mesh	Mesh velocity	vn = −K*nJ_dc
2	Global	Mesh displacement	dx = 0, dy = 0

Boundary Settings: Moving Mesh (ALE)

Using the menu bar, select Multiphysics > Moving Mesh (ALE). Next, using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value in the edit windows as indicated in Table 4.9. See Figures 4.69, 4.70, and 4.71.

Click OK. See Figure 4.72.

Mesh Generation

From the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 1 in the Subdomain selection window. Enter 4e-5 in the Maximum element size edit window. Select "Quad" from the Method drop-down list. See Figure 4.73.

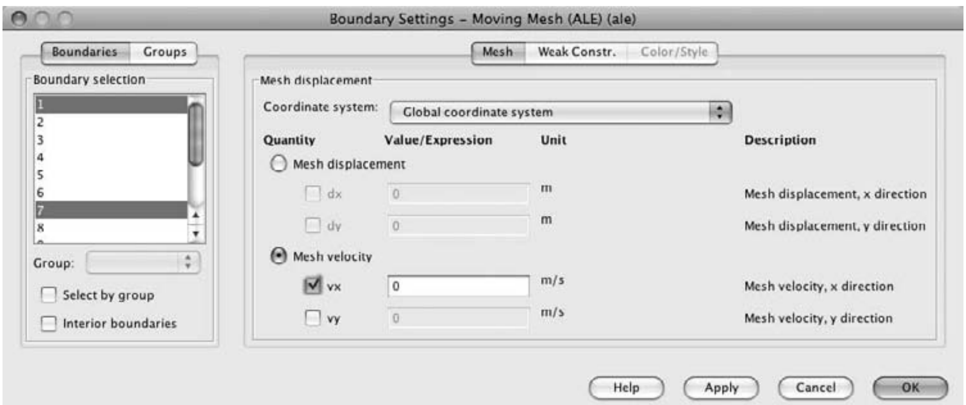


FIGURE 4.69 Boundary Settings (1, 7), Moving Mesh (ALE): boundaries set

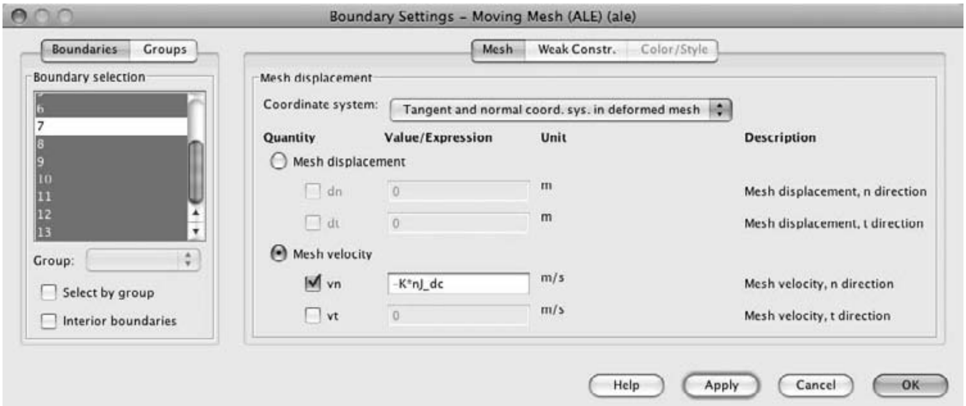


FIGURE 4.70 Boundary Settings (3–6, 8–13), Moving Mesh (ALE): boundaries set

Click the Remesh button, and then click OK. See Figure 4.74. This mesh contains approximately 675 elements.

Solving the Second Variation on the 2D Electrochemical Polishing Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver. Enter 0:0.5:10 in the Times edit window, as shown in Figure 4.75. This instruction causes the Solver to divide the modeling time-space into 20 equal intervals, over the period from 0 to 10 seconds. Click OK.

Using the menu bar, select Solve > Solve Problem.

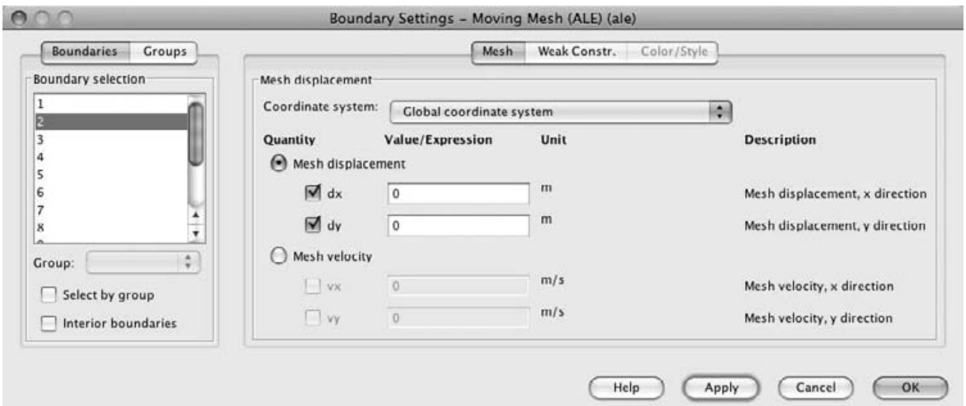


FIGURE 4.71 Boundary Settings (2), Moving Mesh (ALE): boundary set

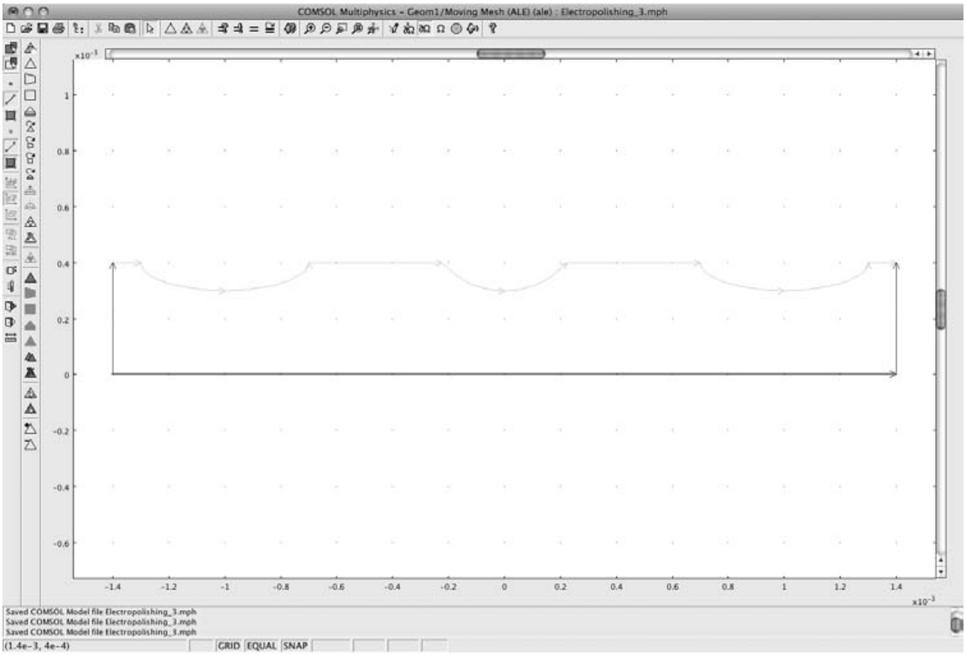


FIGURE 4.72 Boundary Settings, Moving Mesh (ALE): boundaries organized by color (1, 7 = blue; 3–6, 8–13 = green; 2 = red)

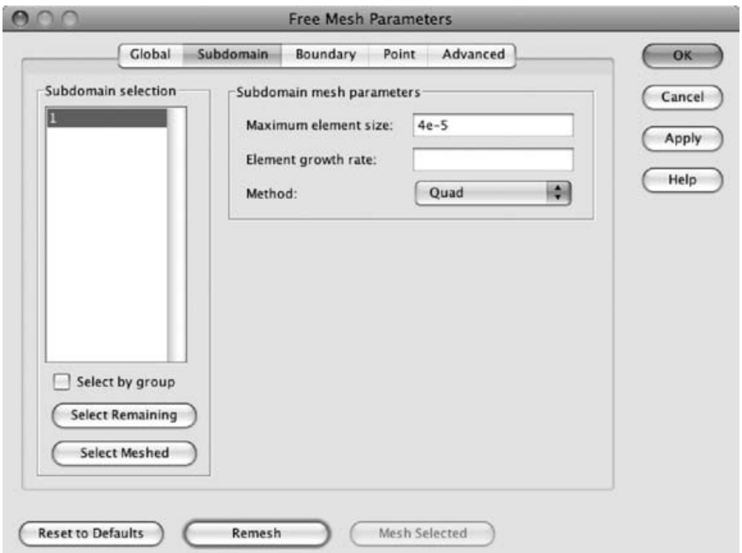


FIGURE 4.73 2D Electrochemical polishing model Free Mesh Parameters window

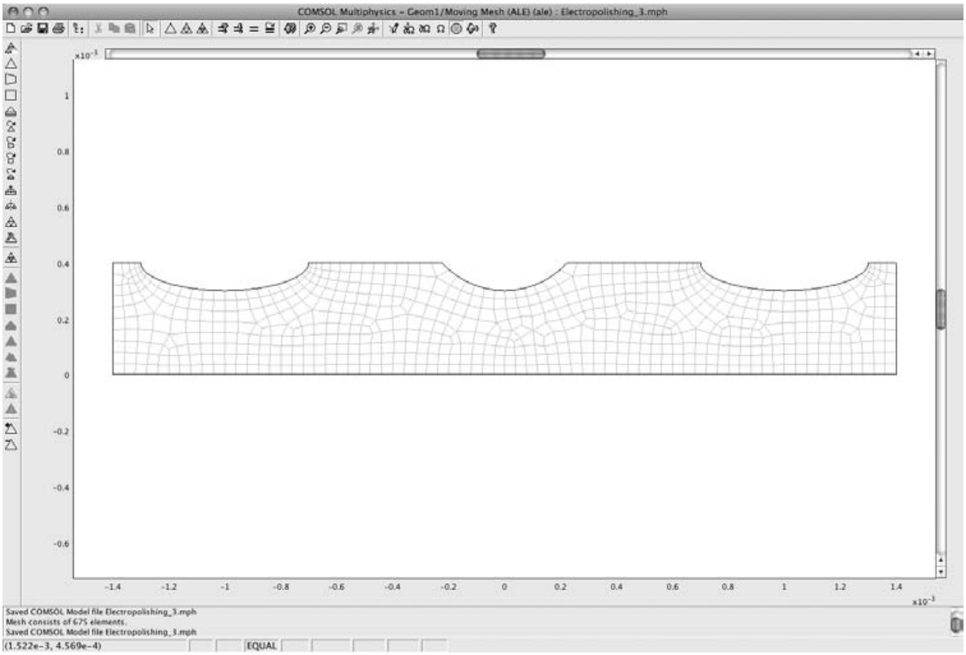


FIGURE 4.74 2D Electrochemical polishing model free mesh (quad)

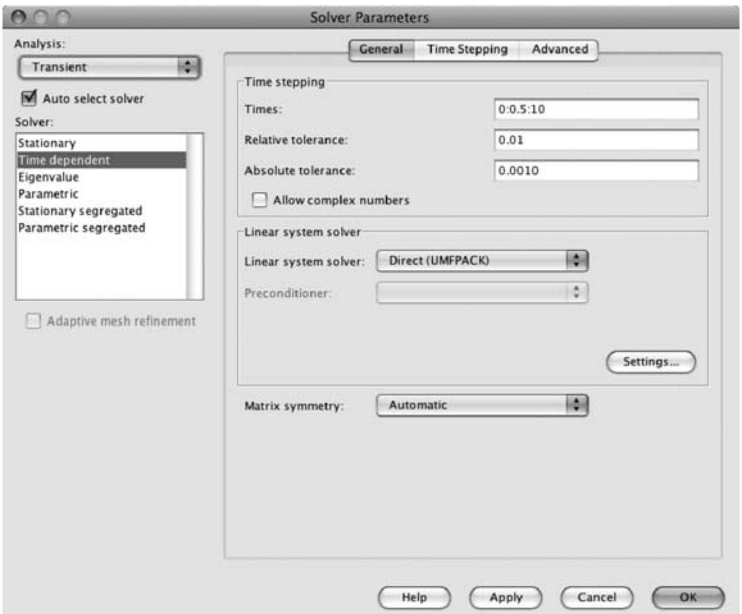


FIGURE 4.75 2D Electrochemical polishing model Solver Parameters window

NOTE In the process of solving this model, using the Moving Mesh Application Mode (ALE), the modeler may occasionally see a warning about an "inverted mesh element." If the solver continues on to a solution, ignore the warning. Such warnings are normal when using the deformed mesh.

If the model does not continue to a solution and the solver displays numerous warnings, then either there is an error in the model or the modeler needs to use the advanced technique called remesh (not discussed in this book).

Postprocessing

Select Postprocessing > Plot Parameters. Click the Surface tab, and verify that the Surface plot check box is checked. From the Predefined quantities drop-down list, select Conductive Media DC (dc) > Total current density, norm. See Figure 4.76. Click OK.

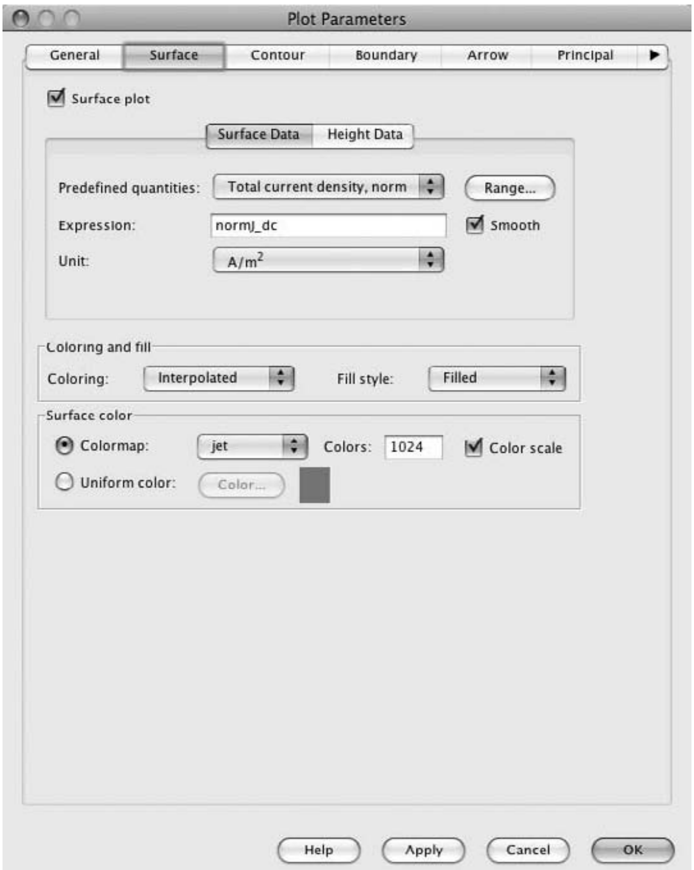


FIGURE 4.76 2D electrochemical polishing model surface Plot Parameters window, total normal current density

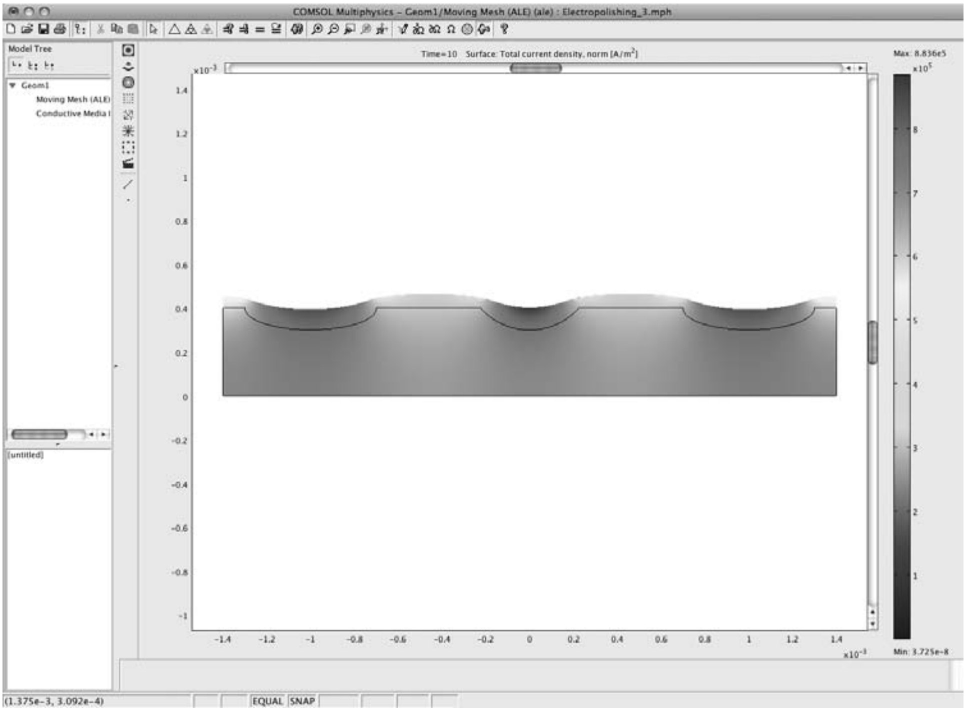


FIGURE 4.77 2D electrochemical polishing model Surface plot window, total normal current density

In Figure 4.77, the model calculation shows that the maximum current density is approximately 8.84e5 A/m^2, in the region of the asperities. Figure 4.77 also shows that the normal current density (J_n) concentrated in the region of the asperities is approximately 1.5 times the normal current density elsewhere on the electrode surface. As a result, the removal rate of the electrode material will be approximately 1.5 times as high.

To see the change in the position of the electrode surface and the relative removal of material from the asperities, select Postprocessing > Plot Parameters. Click the Surface tab, and verify that the Surface plot check box is checked. From the Predefined quantities drop-down list, select Moving Mesh (ALE) (ale) > y-displacement. See Figure 4.78.

Click OK. Figure 4.79 shows the displacement of the electrode surface in the *y*-direction (dy_ale) after 10 seconds of electropolishing.

Postprocessing Animation

This solution to the 2D electrochemical polishing model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, Click the Animate tab.

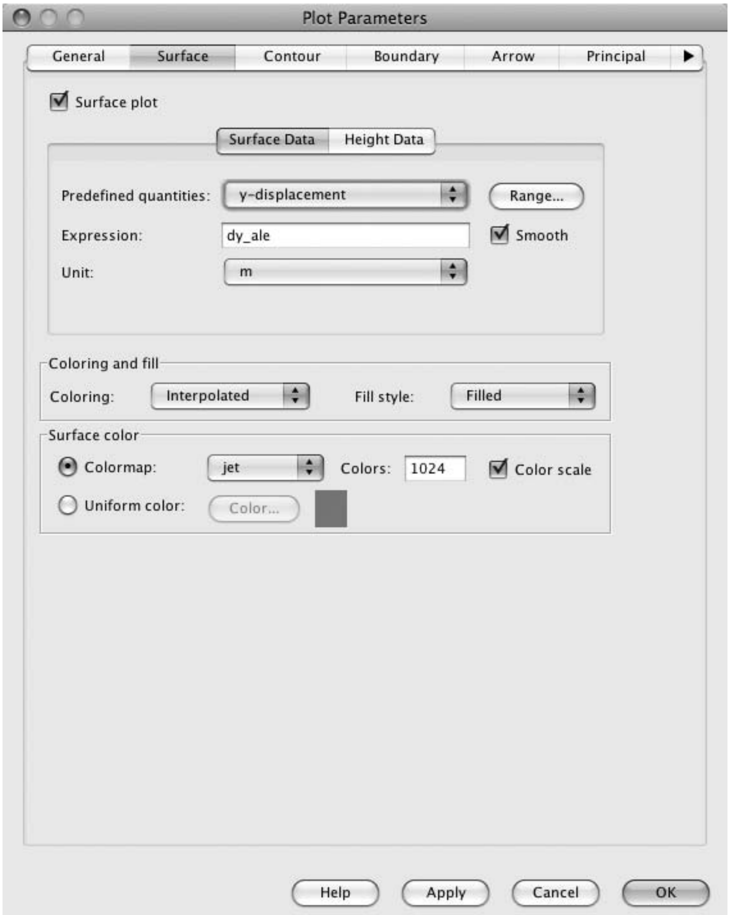


FIGURE 4.78 2D electrochemical polishing model surface Plot Parameters window: Moving Mesh (ALE) (ale), *y*-direction (dy_ale)

On the Animate page, select all the solutions in the Stored output times window (see Figure 4.80). Click the Start Animation button. Save this 2D electrochemical polishing model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4_EP_3.avi that was supplied with this book.

2D Electrochemical Polishing Models: Summary and Conclusions

The models presented in this section have introduced the following new concepts: two-dimensional modeling (2D), deformed mesh—Moving Mesh (ALE), transient analysis, Conductive Media DC, vector dot product current (K*nJ_dc), triangular mesh, free mesh parameters, subdomain mesh, maximum element size, and quadrilateral

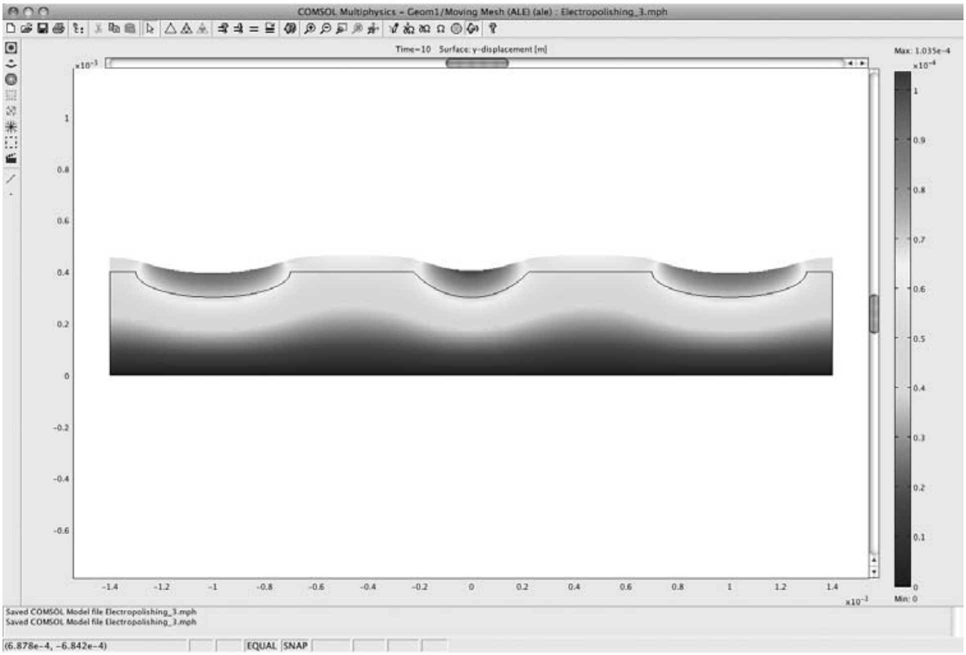


FIGURE 4.79 2D electrochemical polishing model Surface plot window: Moving Mesh (ALE) (ale), *y*-direction (dy_ale)

mesh (quad). The 2D electrochemical polishing model is a powerful tool that can be used to model surface smoothing for diverse projects (e.g., microscope samples, precision metal parts, medical equipment and tools, large and small metal drums, thin analytical samples, vacuum chambers). A comparison of the calculated results for the three electropolishing models is shown in Table 4.10.

The differences between the calculations for the tested models are in the range of a few percentage points. It is left to the modeler to explore other differences between the models by varying the parameters, as suggested in the exercises at the end of this chapter.

Table 4.10 Electropolishing Modeling Results Summary

Model	Asperities	Mesh	Peak J_n	ΔJ_n(%)	dy	Δdy (%)
EP_1	1	Triangular	9.12e5	—	1.08e-4	—
EP_2	1	Quad	8.87e5	~2.7	1.04e-4	~3.7
EP_3	3	Quad	8.83e5	~3.2	1.04e-4	~3.7

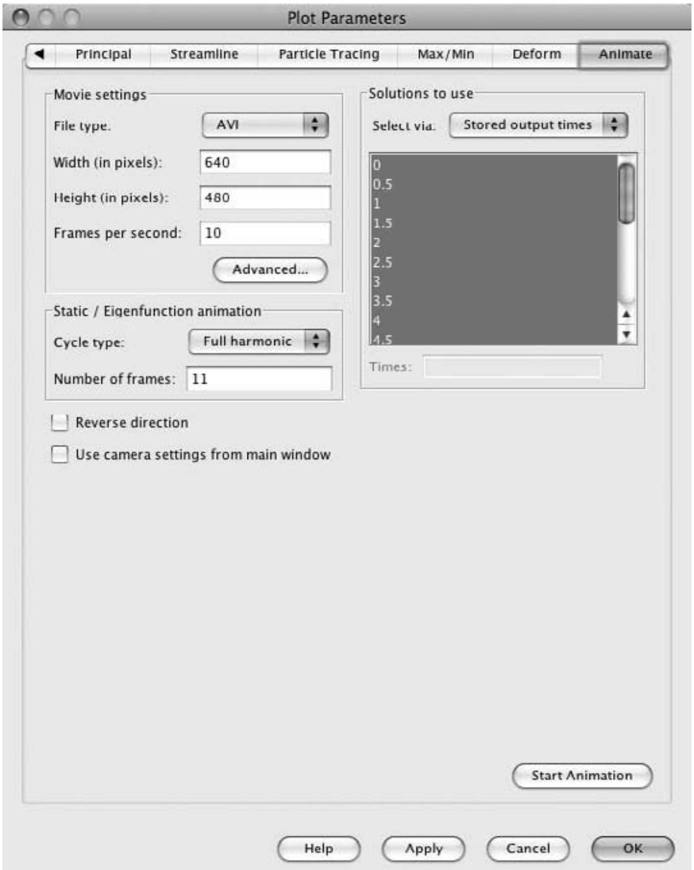


FIGURE 4.80 2D electrochemical polishing model animation Plot Parameters window

2D Hall Effect Model Considerations

In 1827, Georg Ohm published[9] his now fundamental and famous Ohm's law:

$$I = \frac{V}{R} \tag{4.4}$$

where I = current in amperes
V = potential difference in volts
R = resistance in ohms

See Figure 4.81.

As useful as Ohm's law is, it is basically phenomenological. To more fully understand conduction in homogeneous, isotropic solid materials, the calculations

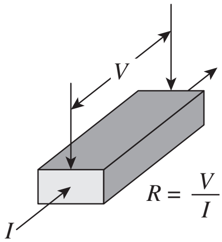


FIGURE 4.81 Ohm's law

need to be expanded until they reflect the behavior (motion) of the fundamental charged particles (electrons, holes).

NOTE In solid materials (e.g., metals, semiconductors), there are three potential mobile carriers of charge: electrons (−), holes (+), and ions (charge sign can be either + or −, depending on the type of ion). Ions in a solid typically have a very low mobility (pinned in position) and, therefore, contribute little to the observed current flow in most solids. Ion flow will not be considered here.

In metals, due to the underlying physical and electronic structure, electrons are the only carrier. In semiconductors (e.g., Si, Ge, GaAs, InP), either electrons or holes (the absence of an electron) can exist as the primary carrier types. The density of each carrier type (electrons, holes) is determined by the electronic structure of the host material (e.g., Si, Ge, SiGe) and the density and distribution of any foreign impurity atoms (e.g., As, P, N, Al) within the host solid material. For further information on the nature of solids and the behavior of impurity atoms in a host matrix, see works by Kittel[10] and Sze.[11]

The resistance of a homogeneous, isotropic solid material R is defined as follows:

$$R = \frac{\rho L}{A} \tag{4.5}$$

where ρ = resistivity in ohm-meters (Ω-m)
L = length of sample in meters (m)
A = cross-sectional area of sample in meters squared (m^2)

See Figure 4.82.

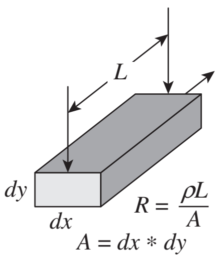


FIGURE 4.82 Resistance

The resistivity of a homogeneous, isotropic solid material is defined as follows:[12]

$$\sigma \equiv \frac{1}{\rho} = n_e|e|\mu_e + n_h|e|\mu_h \tag{4.6}$$

where
ρ = resistivity ohm-meters (Ω-m)
σ = conductivity in siemens per meter (S/m)
n_e = electron density in electrons per cubic meter (N_e/m^3)
n_h = hole density in holes per cubic meter (N_h/m^3)
$|e|$ = absolute value of the charge on an electron (hole) in coulombs (C)
μ_e = electron mobility in meters squared per volt-second ($m^2/(V*s)$)
μ_h = hole mobility in meters squared per volt-second ($m^2/(V*s)$)

The Hall effect[13] was discovered by Edwin Hall in 1879[14] through measurements on the behavior of currents in thin gold foils, in the presence of a magnetic field. The magnetic field introduced into the current flow region of the solid in the Hall effect measurements effectively adds an anisotropic term into the conductivity of a nominally homogeneous, isotropic solid material. The anisotropic conductivity is caused by the magnetic field through the Lorentz force.[15] The Lorentz force produces a proportional, differential voltage/charge accumulation between two surfaces or edges of a conducting material orthogonal to the current flow.

The Lorentz force is

$$\mathbf{F} = q\,(\mathbf{E} + (\mathbf{v} \times \mathbf{B})) \tag{4.7}$$

where
$\mathbf{F}$ = force vector on the charged particle (electron and/or hole)
q = charge on the particle (electron and/or hole)
$\mathbf{E}$ = electric field vector
$\mathbf{v}$ = instantaneous velocity vector of the particle
$\mathbf{B}$ = magnetic field vector

The Hall voltage[16] is

$$V_H = \frac{R_H{*}I{*}B}{t} \tag{4.8}$$

where V_H = Hall voltage
R_H = Hall coefficient
I = current
B = magnetic field
t = thickness of sample

The Hall coefficient (R_H) is

$$R_H = -\frac{r}{n_e e} \tag{4.9}$$

where R_H = Hall coefficient
$r = 1 \leq \times \leq 2$
n_e = density of electrons
e = charge on the electron

NOTE In the Hall effect models presented here, it is assumed that $r = 1$. That assumption is a valid first approximation. For applied development models, the modeler will need to determine experimentally the best approximation for the value of r for the particular material and physical conditions being modeled.

For example, in the case that the charge carrier is a "hole," the minus sign (−) in the equation for the Hall coefficient changes to a plus sign (+). In the case of mixed electron/hole flow, R_H can become zero.

The differential voltage/charge accumulation—Hall voltage (V_H)—that results from the Lorentz force interaction between any currents (electron and/or hole) flowing through that conducting material and the local magnetic field is shown in Figure 4.83.

As can be seen from the introductory material, depending on the characteristics of the material being modeled, the calculation of the Hall effect can be very complex. The Hall coefficient (R_H) varies for different materials and has a predominant functional dependence that involves temperature, carrier type, carrier concentration, carrier mobility, carrier lifetime, and carrier velocity. In a dual-carrier system, such as semiconducting materials (electrons and holes), under the proper conditions, R_H can become equal to zero. Semiconductor sensors, however, are among the most sensitive magnetic field Hall sensors currently manufactured.

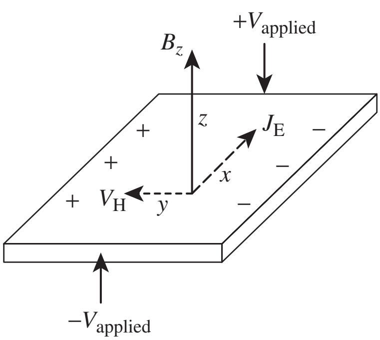


FIGURE 4.83 Hall effect sensor geometry, electron flow

Hall effect sensors are widely available in a large number of geometric configurations. They are typically applied in sensing fluid flow, rotating or linear motion, proximity, current, pressure, and orientation. In the 2D models presented in the remainder of this chapter, several simplifying assumptions will be made that allow the basic physics principles to be demonstrated without excessive complexity.

Owing to the underlying complexity of the Hall effect, the models in this section of Chapter 4 require the use of either the AC/DC Module or the MEMS Module, in addition to the basic COMSOL Multiphysics software. In the first model, only a single carrier conduction system (electrons) will be employed. For ease of modeling, it will be assumed that the system is quasi-static. This model introduces the COMSOL modeling concepts of point constraints and floating contacts.[17]

2D Hall Effect Model

To start building the Hall_Effect_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select AC/DC Module > Statics > Conductive Media DC. Click the Multiphysics button, and then click the Add button. See Figure 4.84.

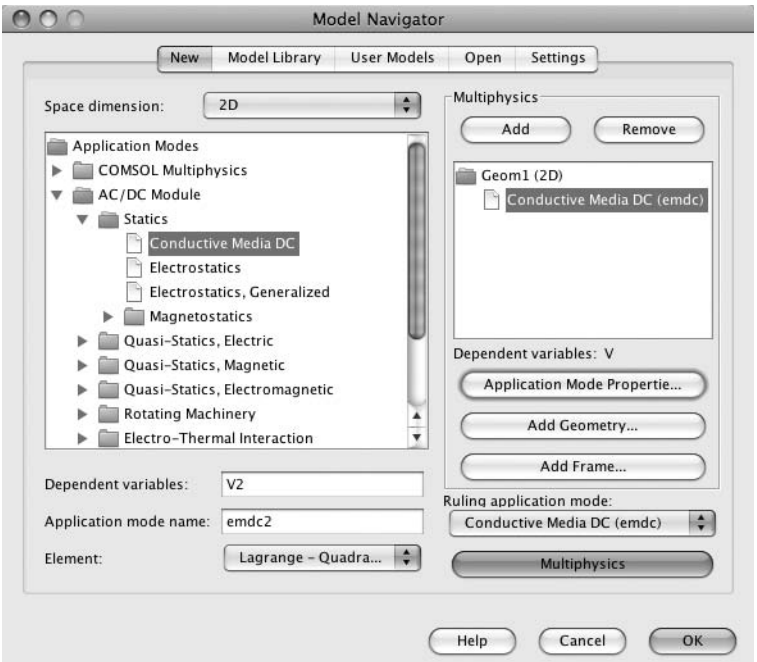


FIGURE 4.84 Multiphysics Model Navigator window

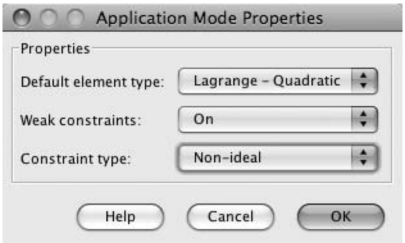


FIGURE 4.85 Application Mode Properties window

Click the Application Mode Properties button. Select "On" from the Weak constraints pull-down list. Select "Non-ideal" from the Constraint type pull-down list. See Figure 4.85. Click OK.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.11; also see Figure 4.86. Click OK.

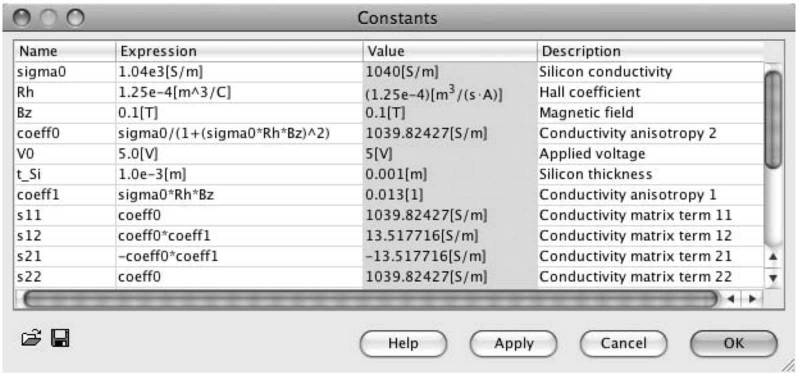
2D Hall Effect Geometry

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 1.8e-2 and height of 6e-3. Select "Base: Corner" x equal to -9e-3 and y equal to -3e-3 in the Rectangle edit window. See Figure 4.87.

Click OK, and then click the Zoom Extents button. See Figure 4.88.

Table 4.11 Constants Edit Window

Name	Expression	Description
sigma0	1.04e3[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	Hall coefficient
Bz	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	Conductivity anisotropy 2
V0	5.0[V]	Applied voltage
t_Si	1.0e-3[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	Conductivity anisotropy 1
s11	coeff0	Conductivity matrix term 11
s12	coeff0*coeff1	Conductivity matrix term 12
s21	-coeff0*coeff1	Conductivity matrix term 21
s22	coeff0	Conductivity matrix term 22



Name	Expression	Value	Description
sigma0	1.04e3[S/m]	1040[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	(1.25e-4)[m^3/(s·A)]	Hall coefficient
Bz	0.1[T]	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	1039.82427[S/m]	Conductivity anisotropy 2
V0	5.0[V]	5[V]	Applied voltage
t_Si	1.0e-3[m]	0.001[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	0.013[1]	Conductivity anisotropy 1
s11	coeff0	1039.82427[S/m]	Conductivity matrix term 11
s12	coeff0*coeff1	13.517716[S/m]	Conductivity matrix term 12
s21	-coeff0*coeff1	-13.517716[S/m]	Conductivity matrix term 21
s22	coeff0	1039.82427[S/m]	Conductivity matrix term 22

FIGURE 4.86 2D Hall_Effect_1 model Constants edit window

In this model, points will be added to the boundary of the rectangle to define the location of the edges of the floating contacts. Using the menu bar, select Draw > Specify Objects > Point. In the Draw > Specify Objects > Point edit window, individually create each of the points shown in Table 4.12 by selecting the window, entering the data, and then clicking OK. The final rectangle with all four points is shown in Figure 4.89.

NOTE The points are added to the boundary of the rectangle so that the edges of the floating contacts are precisely defined.

2D Hall Effect Subdomain Settings

Using the menu bar, select Physics > Subdomain Settings > Subdomain 1 (the only choice). Enter t_Si in the d (Thickness) edit window. Verify that "Conductivity" is selected in the Conductivity relation pull-down list. Click in the Electric conductivity

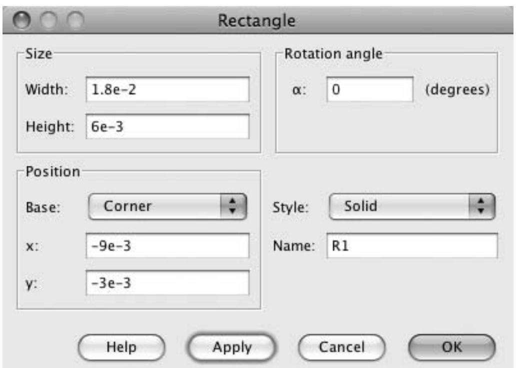


FIGURE 4.87 2D Hall_Effect_1 model Rectangle edit window

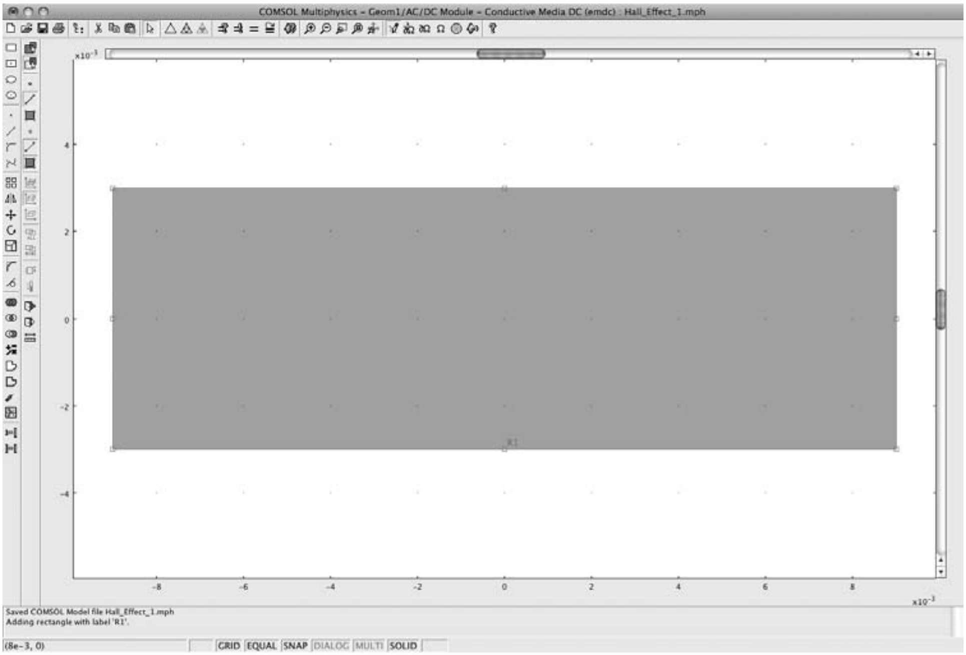


FIGURE 4.88 2D Hall_Effect_1 model rectangle geometry

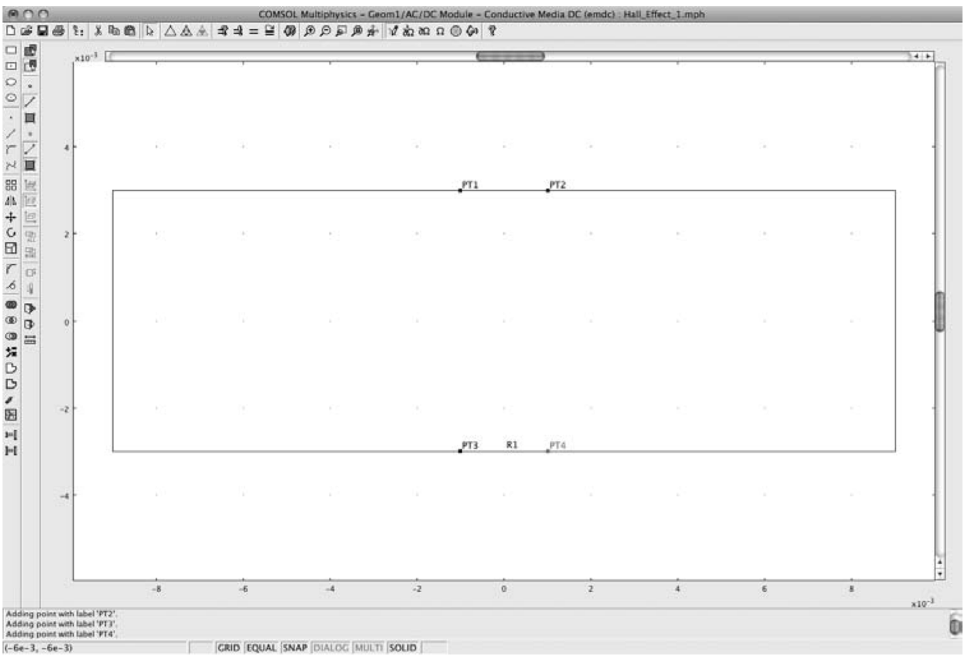


FIGURE 4.89 2D Hall_Effect_1 model rectangle geometry with points

Table 4.12 Points Edit Window

Point Number	*x* Location	*y* Location
1	-1e-3	3e-3
2	1e-3	3e-3
3	-1e-3	-3e-3
4	1e-3	-3e-3

Table 4.13 Matrix Elements Edit Window

Matrix Element Number	Value
11	s11
12	s12
21	s21
22	s22

edit window. Select "Anisotropic-full" from the Conductivity type pull-down list. Enter the matrix elements as shown in Table 4.13; see Figure 4.90.

NOTE These matrix elements define the anisotropic coupling of the magnetic field and the current flowing in the silicon sample.

Close the Conductivity Matrix edit window by clicking on the Subdomain Settings window. After the Conductivity Matrix window closes, the matrix elements will be as shown in the Conductivity edit window on the Subdomain Settings page in Figure 4.91. Click OK.

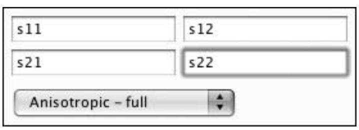


FIGURE 4.90 2D Hall_Effect_1 model conductivity matrix elements

Table 4.14 Boundary Settings

Boundary Number	Condition	Group Index	Source Current/Potential	Figure
1	Ground	—	—	4.92
2, 3, 6, 7	Electric insulation	—	—	4.93
4	Floating potential	2	0	4.94
5	Floating potential	1	0	4.95
8	Electric potential	—	V0	4.96

2D Hall Effect Boundary Settings

Using the menu bar, select Physics > Boundary Settings. Enter the boundary settings as shown in Table 4.14. See Figures 4.92, 4.93, 4.94, 4.95, and 4.96.

NOTE The addition of the group index designation decouples the two floating contacts from each other. Failure to insert a different group index number for each floating contact couples (mathematically short-circuits) the contacts together.

Click the Weak Constr. tab. Verify that the Use weak constraints check box is checked. See Figure 4.97.

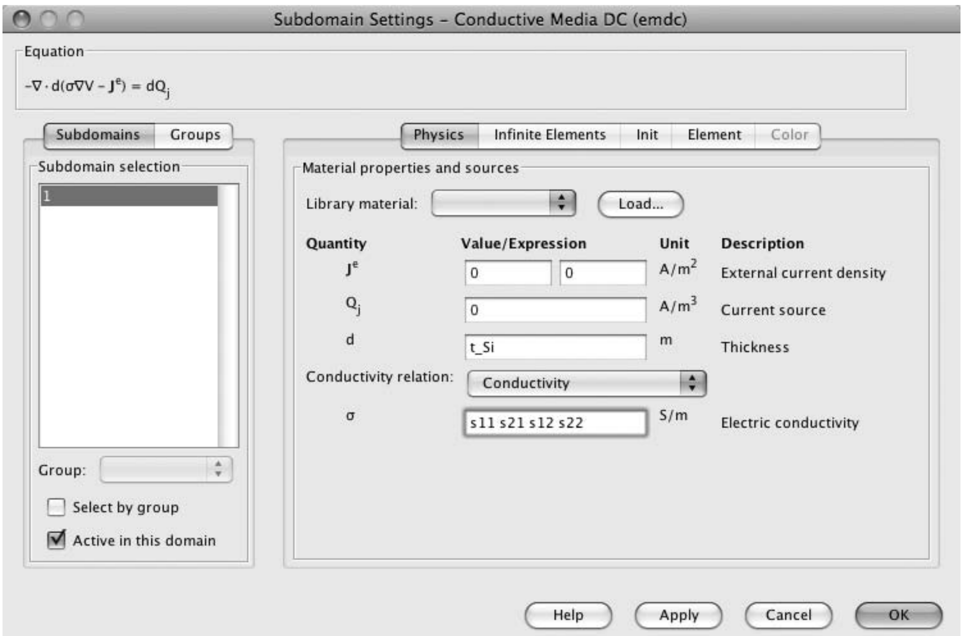


FIGURE 4.91 2D Hall_Effect_1 model Subdomain Settings

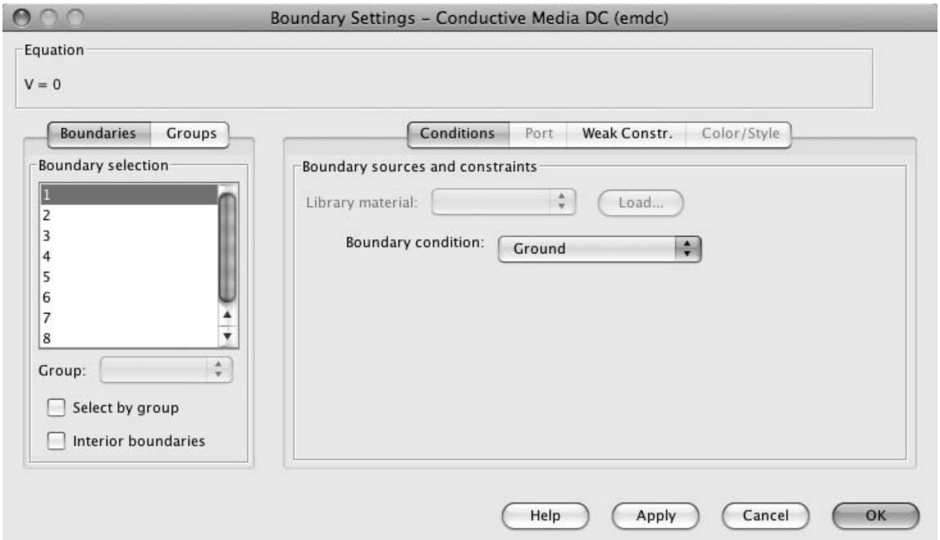


FIGURE 4.92 2D Hall_Effect_1 model Boundary Settings (1)

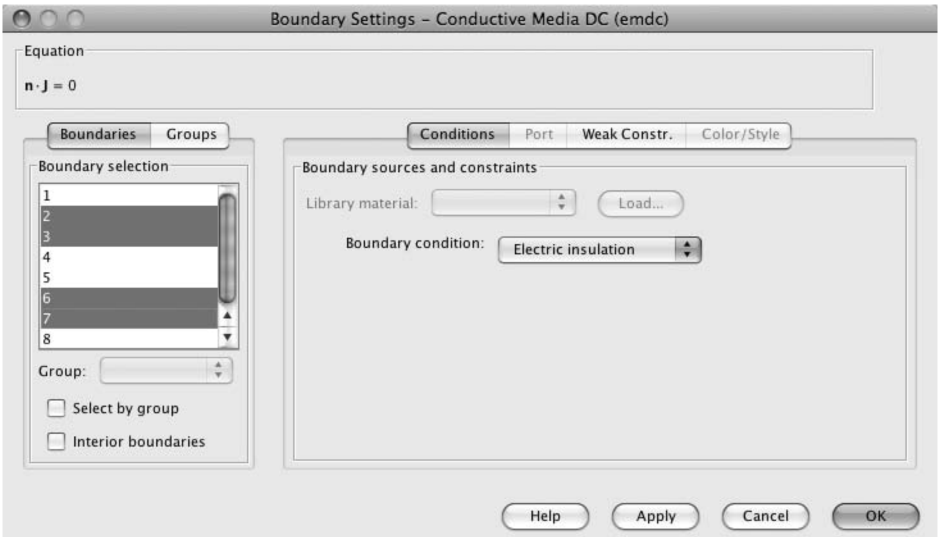


FIGURE 4.93 2D Hall_Effect_1 model Boundary Settings (2, 3, 6, 7)

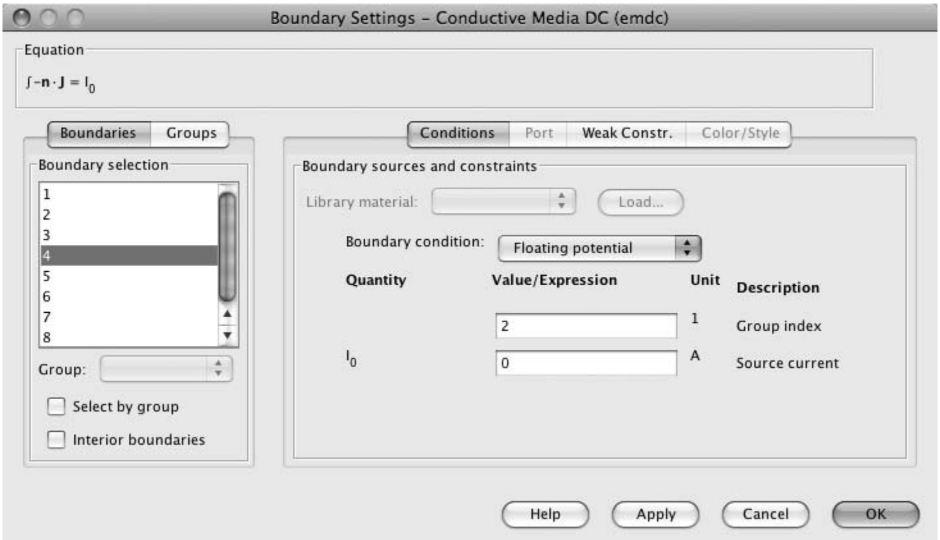


FIGURE 4.94 2D Hall_Effect_1 model Boundary Settings (4)

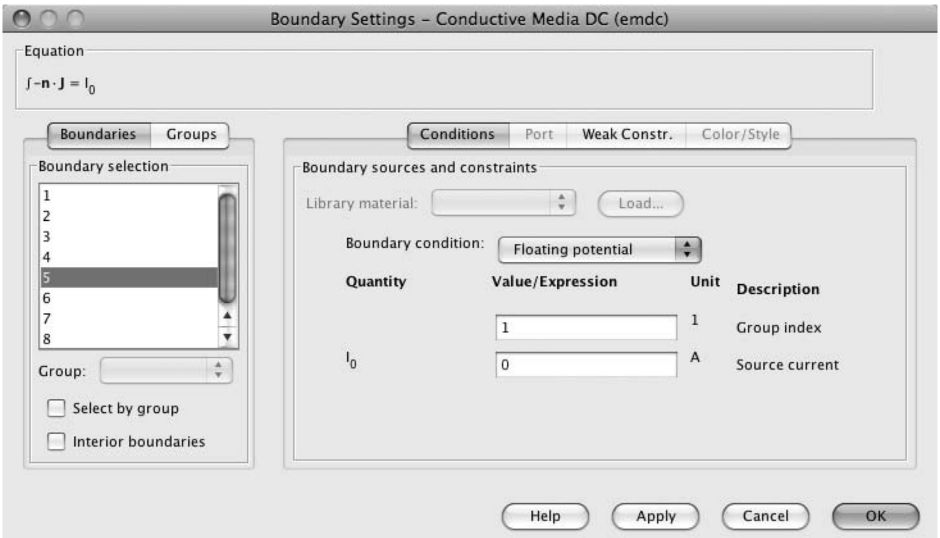


FIGURE 4.95 2D Hall_Effect_1 model Boundary Settings (5)

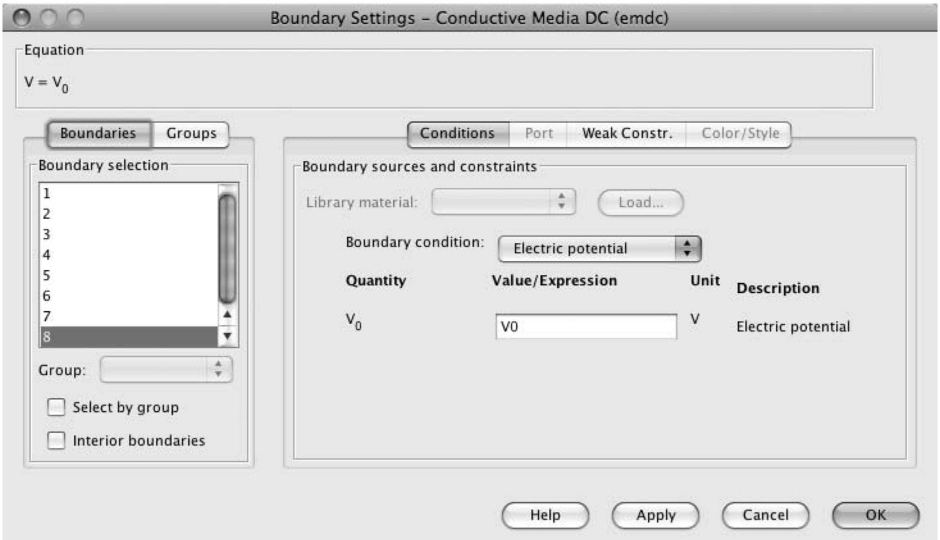


FIGURE 4.96 2D Hall_Effect_1 model Boundary Settings (8)

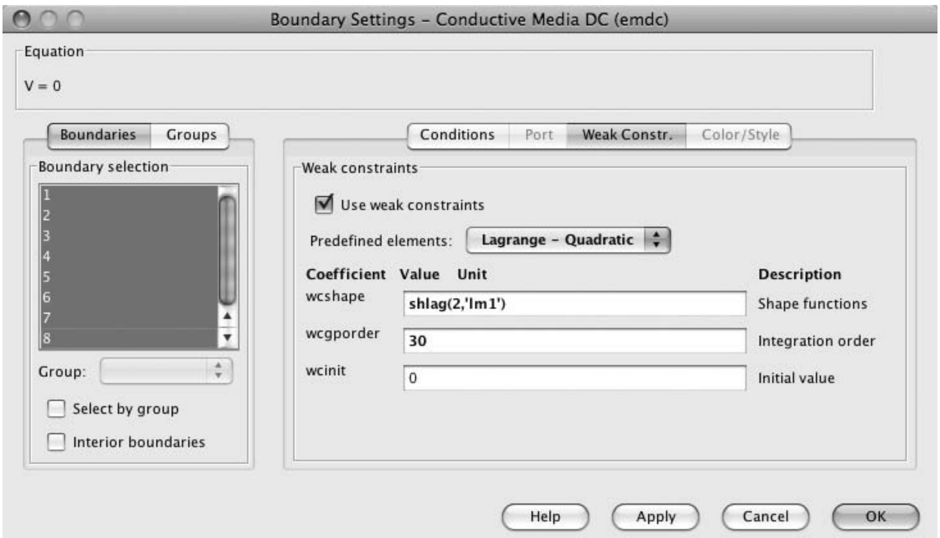


FIGURE 4.97 2D Hall_Effect_1 model Boundary Settings, Weak Constr. page

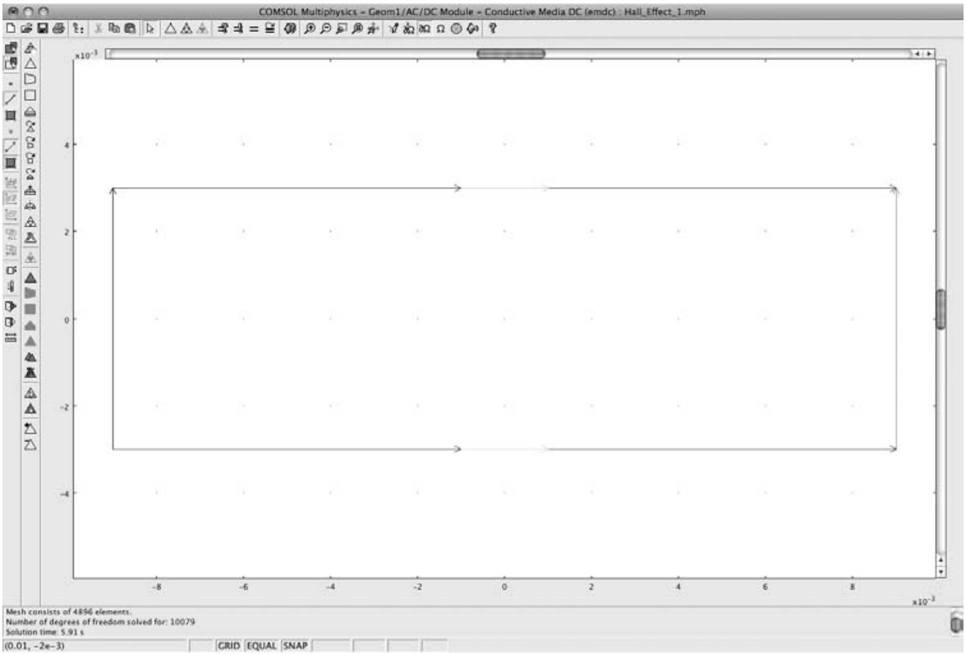


FIGURE 4.98 2D Hall_Effect_1 model boundary settings, final configuration

Click OK. The final configuration of the boundary settings is shown in Figure 4.98.

2D Hall Effect Mesh Generation

Using the menu bar, select Mesh > Initialize Mesh. Click the Refine Mesh button twice. See Figure 4.99.

Solving the 2D Hall Effect Model

Using the menu bar, select Solve > Solver Parameters. In the Solver selection window, select "Parametric." In the Parameter names edit window, enter Bz. In the Parameter values edit window, enter 0:0.1:2.0. See Figure 4.100. Click OK.

NOTE The Parametric Solver is chosen, in this case, so that the modeler can solve the Hall_Effect_1 model quasi-statically over a range of Bz. This allows the modeler to see solutions for a wide range of magnetic field values.

Using the menu bar, select Solve > Solve Problem.

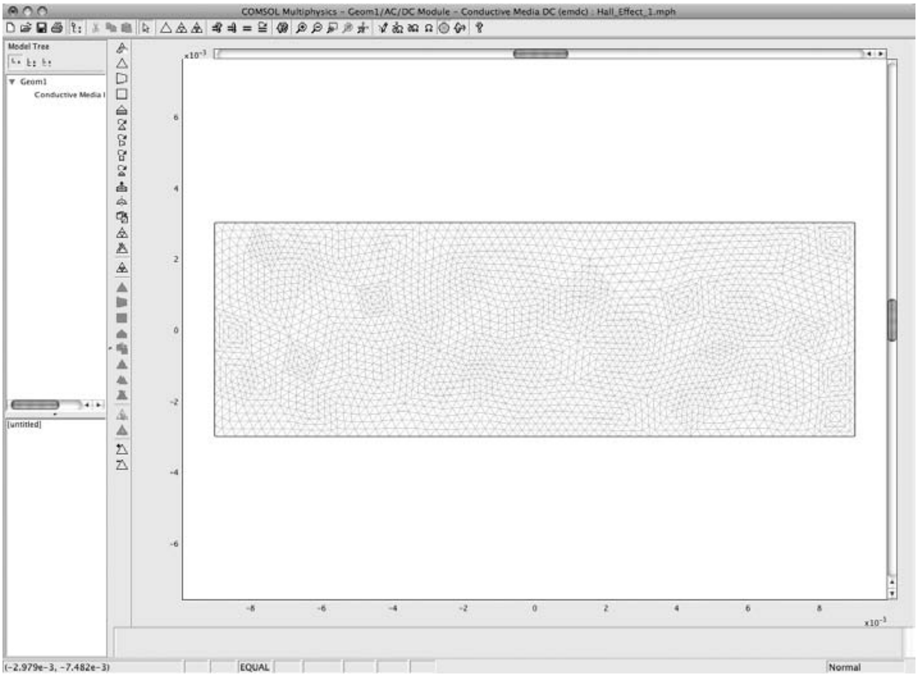


FIGURE 4.99 2D Hall_Effect_1 model mesh

Postprocessing

The default plot is a 2D surface plot of the voltage distribution at the highest value of the magnetic field (Bz = 2 tesla). See Figure 4.101.

More detailed information is displayed by adding contour lines. Using the menu bar, select Postprocessing > Plot Parameters. Click the Contour tab. Place a check mark in the Contour plot check box. Click the Uniform color radio button. Click the Color select button, and select "Black." Click OK. See Figure 4.102.

The Hall effect voltage (V_H) can be seen as the voltage difference (color difference) between the top electrode and the bottom electrode, as shown in Figure 4.103.

NOTE There are two methods by which the voltage difference between the upper and lower surfaces can be determined in Figure 4.103. The first is by the color difference, as indicated by the color bar on the right side of the plot. The second is by the incremental position of the contour lines. If the voltage is constant in the vertical direction, the contour line will be straight and vertical. If the voltage changes, that change is reflected in the shape of the contour line.

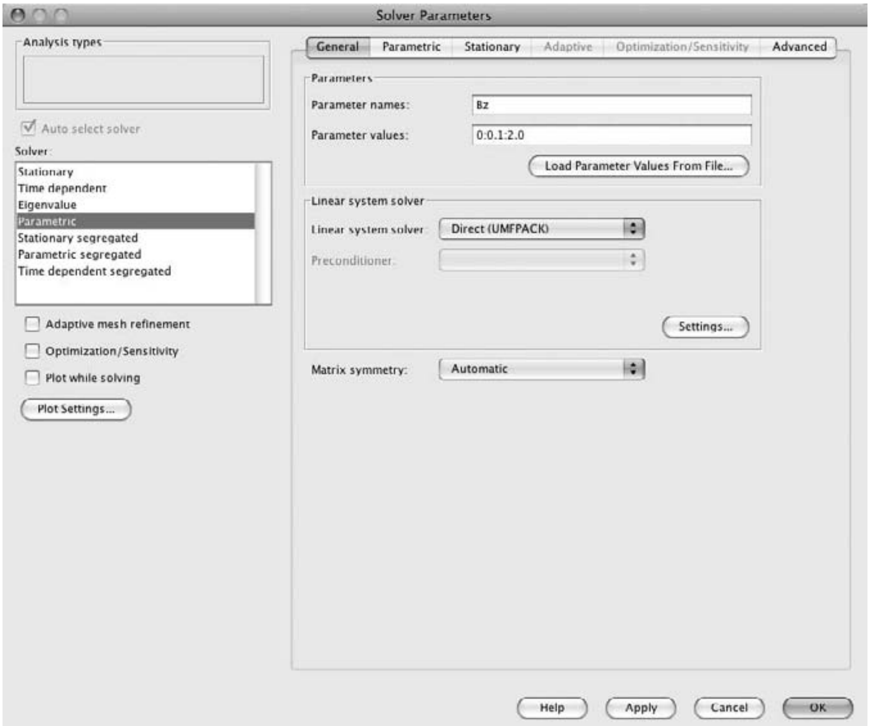


FIGURE 4.100 2D Hall_Effect_1 model Solver Parameters window

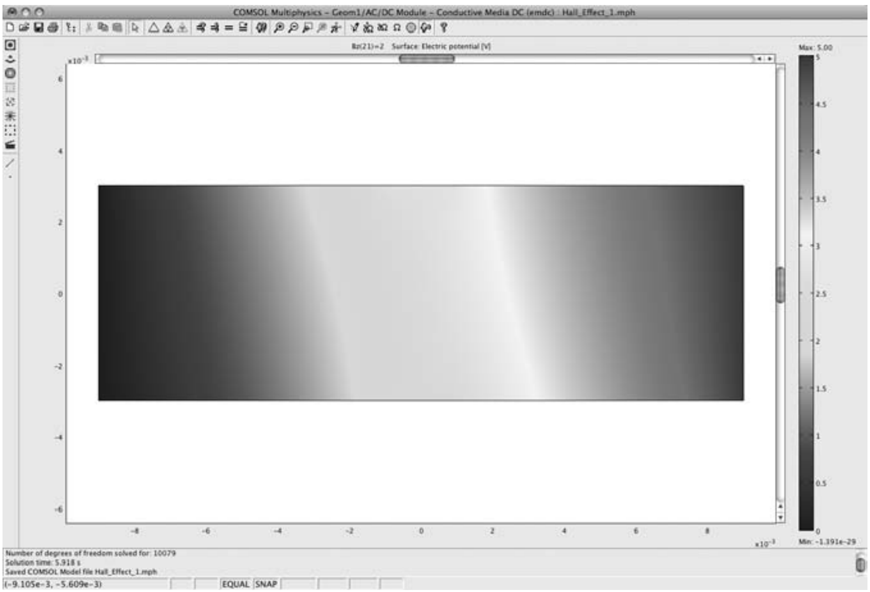


FIGURE 4.101 2D Hall_Effect_1 model default surface voltage distribution plot

FIGURE 4.102 2D Hall_Effect_1 model Plot Parameters window, Contour Data page

The exact voltage difference at any point in the model can be determined by creating a cross-section plot. Using the menu bar, select Postprocessing > Cross-Section Plot Parameters. Select the 2T solution in the Solutions to use window. See Figure 4.104.

Click the Line/Extrusion tab. Enter the coordinates shown in Table 4.15 on the Cross-Section Plot Parameters page. See Figure 4.105.

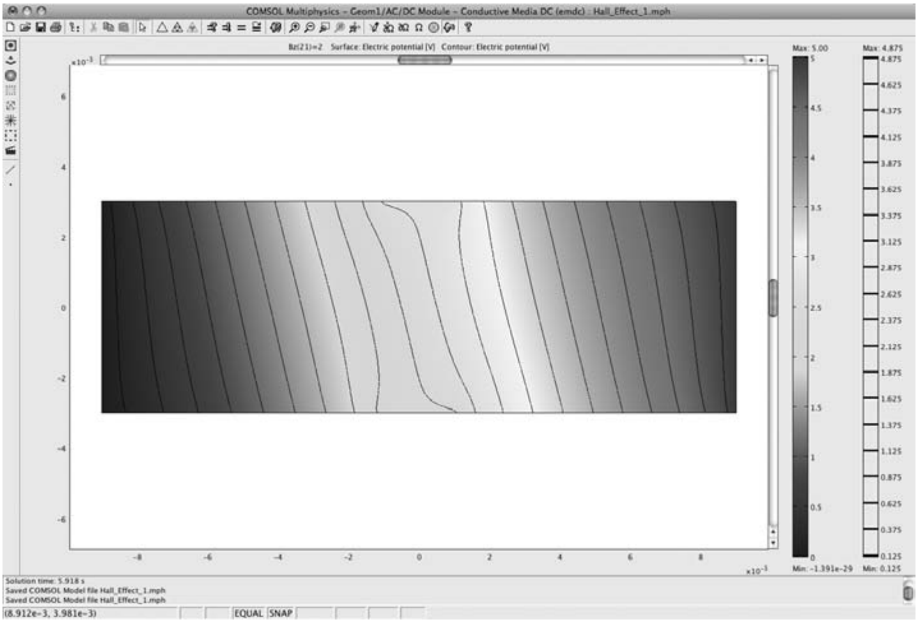


FIGURE 4.103 2D Hall_Effect_1 model surface voltage distribution plot (2T), with contour lines

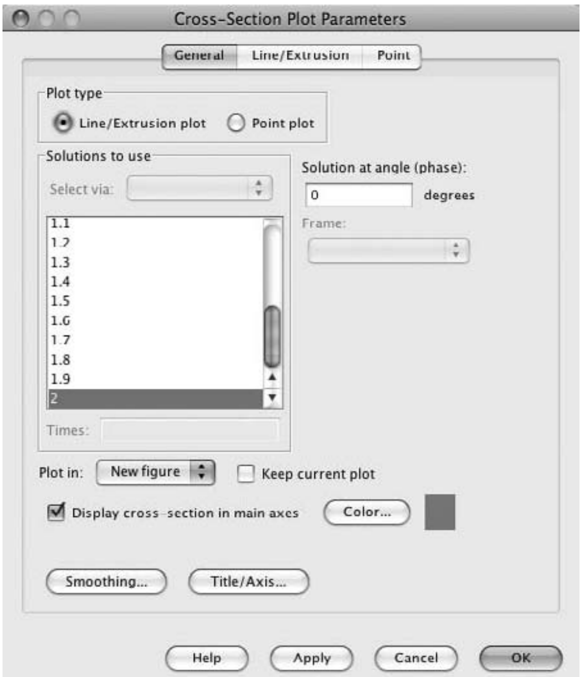


FIGURE 4.104 2D Hall_Effect_1 model Cross-Section Plot Parameters, General page

Table 4.15 Cross-Section Line Data Edit Window

Line Data	Value
x0	0e-3
x1	0e-3
y0	-3e-3
y1	3e-3

Click OK. Figure 4.106 shows the voltage difference (V_H) between the electrode (top) and the modeled Si sample (bottom), for the line $x = 0$. In this case $V_H = 0.340$ volt ($V_{high} - V_{low} = 0.340$ V).

Postprocessing Animation

This solution to the 2D Hall_Effect_1 model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters.

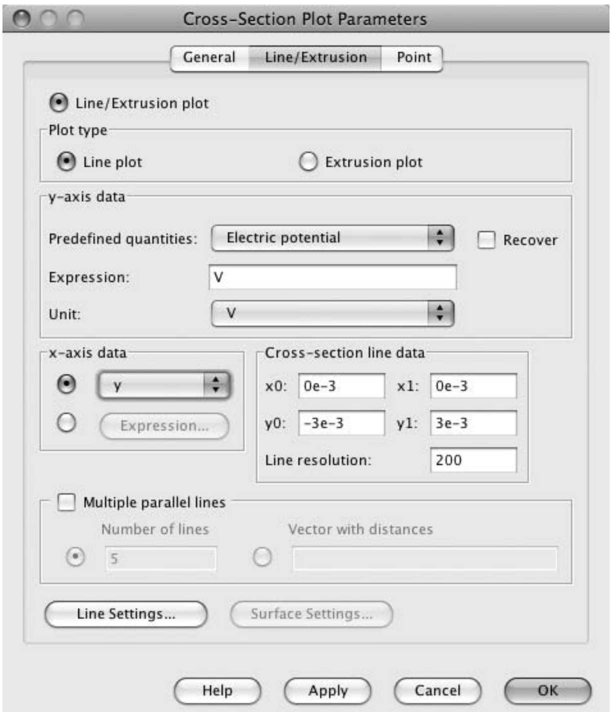


FIGURE 4.105 2D Hall_Effect_1 model Cross-Section Plot Parameters, Line/Extrusion page

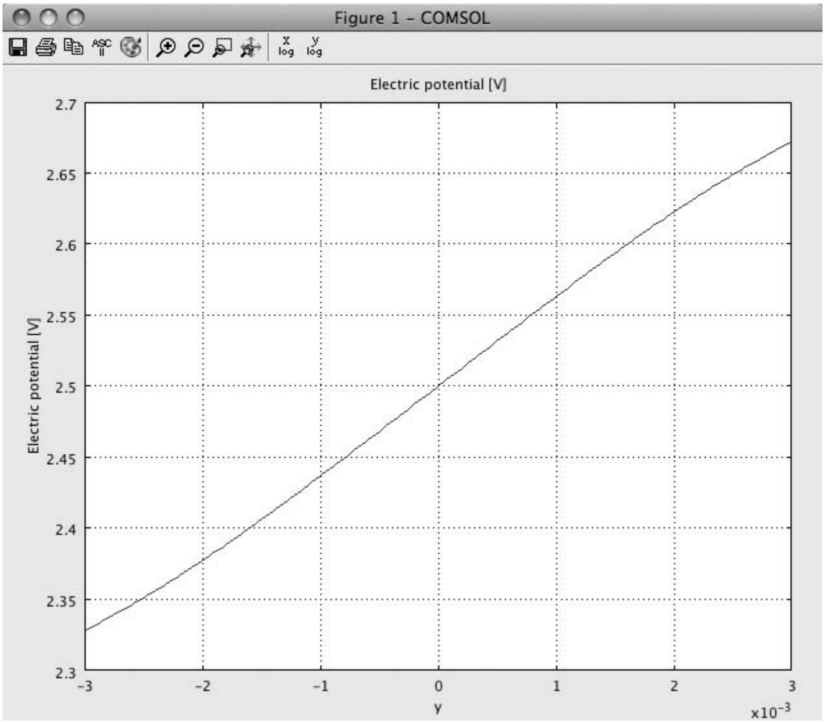


FIGURE 4.106 2D Hall_Effect_1 model plot V_H

Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 4.107). Click the Start Animation button. Save this 2D Hall effect model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4_HE_1.avi that was supplied with this book.

First Variation on the 2D Hall Effect Model

This model reflects a closer approach to the construction of a specimen as would be constructed from a silicon wafer. In this model, both Si end contacts and side contacts have been added, as would be the case for a fabricated Si sample.

To start building the Hall_Effect_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select AC/DC Module > Statics > Conductive Media DC. Click the Multiphysics button, and then click the Add button. See Figure 4.108.

Click the Application Mode Properties button. Select "On" from the Weak constraints pull-down list. Select "Non-ideal" from the Constraint type pull-down list. See Figure 4.109. Click OK.

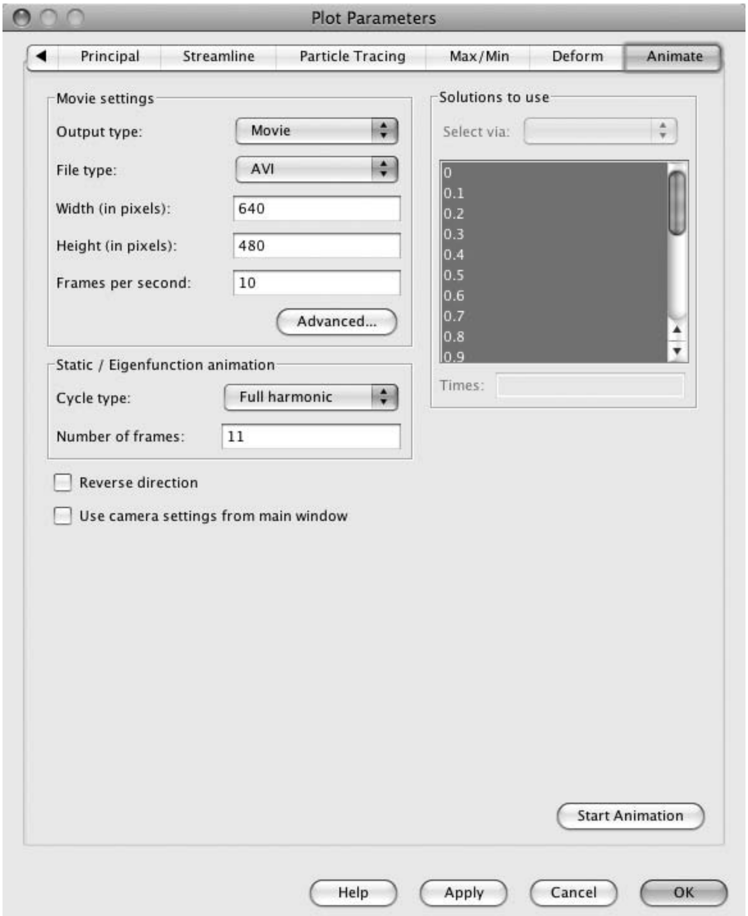


FIGURE 4.107 2D Hall_Effect_1 model animation Plot Parameters window

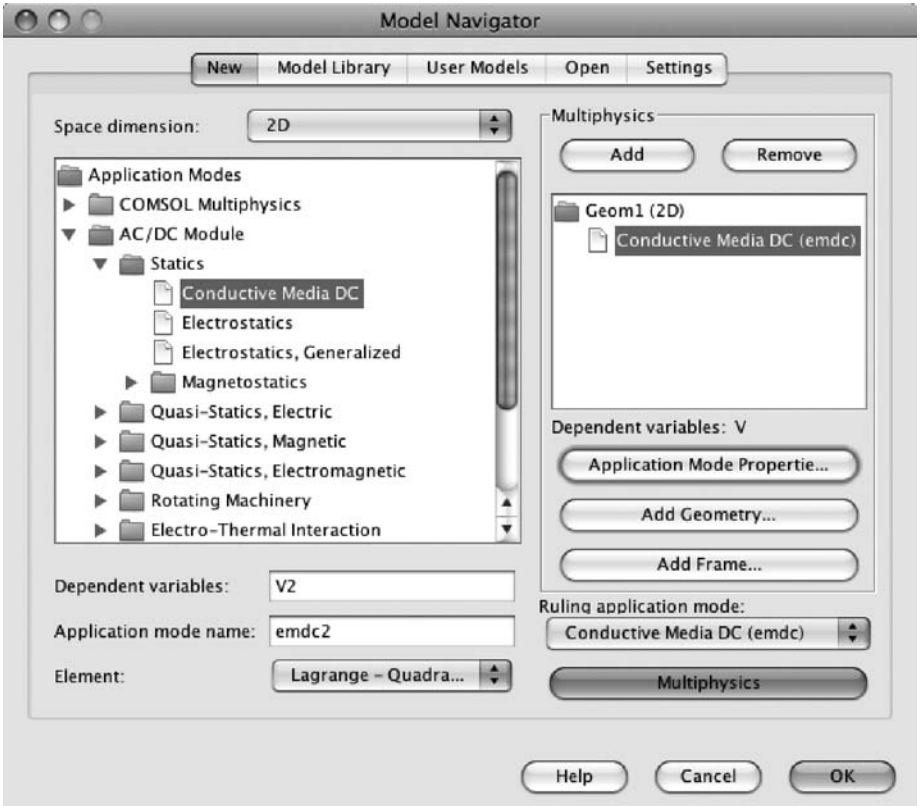


FIGURE 4.108 Multiphysics Model Navigator window

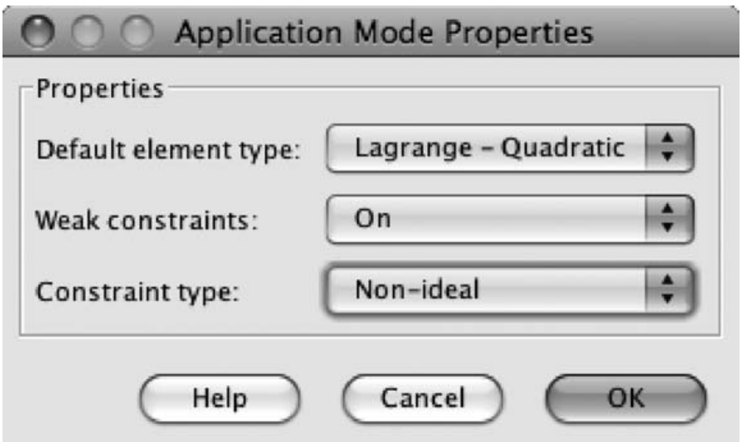
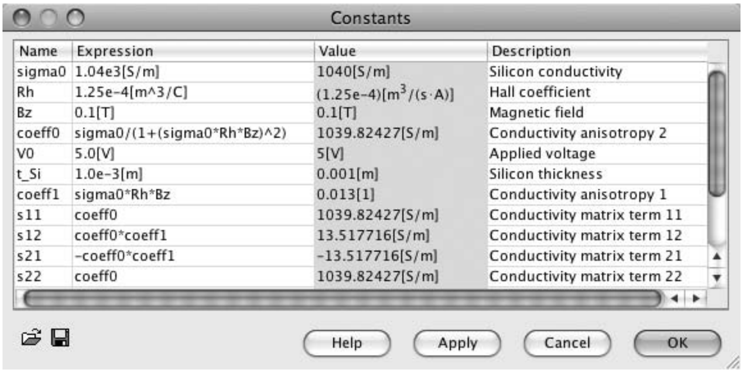


FIGURE 4.109 Application Mode Properties window



Name	Expression	Value	Description
sigma0	1.04e3[S/m]	1040[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	(1.25e-4)[m^3/(s·A)]	Hall coefficient
Bz	0.1[T]	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	1039.82427[S/m]	Conductivity anisotropy 2
V0	5.0[V]	5[V]	Applied voltage
t_Si	1.0e-3[m]	0.001[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	0.013[1]	Conductivity anisotropy 1
s11	coeff0	1039.82427[S/m]	Conductivity matrix term 11
s12	coeff0*coeff1	13.517716[S/m]	Conductivity matrix term 12
s21	-coeff0*coeff1	-13.517716[S/m]	Conductivity matrix term 21
s22	coeff0	1039.82427[S/m]	Conductivity matrix term 22

FIGURE 4.110 2D Hall_Effect_2 model Constants edit window

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.16; also see Figure 4.110. Click OK.

2D Hall Effect Geometry

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 1.8e-2, and a height of 6e-3. Select "Base: Corner" x and set equal to -9e-3 and y equal to -3e-3 in the Rectangle edit window. See Figure 4.111.

Click OK, and then the click the Zoom Extents button. See Figure 4.112.

Table 4.16 Constants Edit Window

Name	Expression	Description
sigma0	1.04e3[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	Hall coefficient
Bz	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	Conductivity anisotropy 2
V0	5.0[V]	Applied voltage
t_Si	1.0e-3[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	Conductivity anisotropy 1
s11	coeff0	Conductivity matrix term 11
s12	coeff0*coeff1	Conductivity matrix term 12
s21	-coeff0*coeff1	Conductivity matrix term 21
s22	coeff0	Conductivity matrix term 22

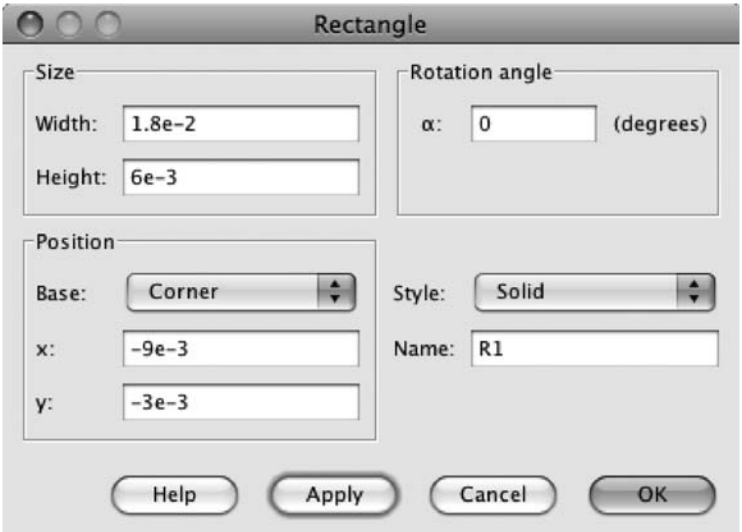


FIGURE 4.111 2D Hall_Effect_2 model Rectangle edit window

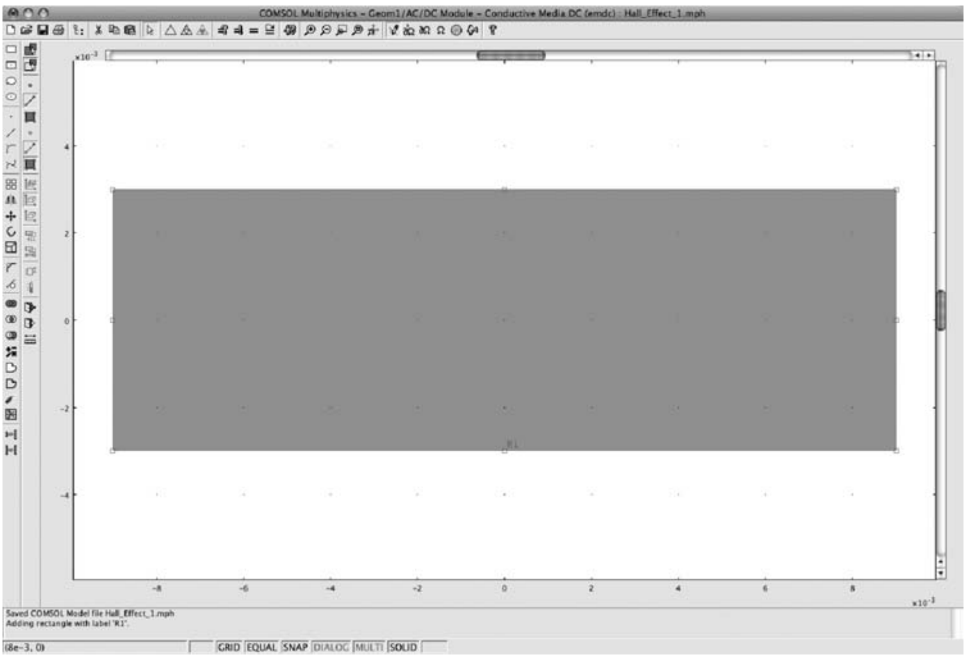


FIGURE 4.112 2D Hall_Effect_2 model rectangle geometry

Table 4.17 Rectangle Edit Window

Rectangle Number	Width	Height	Base	*x* Location	*y* Location
1	2e-3	1e-3	Corner	-1e-3	3e-3
2	2e-3	1e-3	Corner	-1e-3	-4e-3
3	1e-3	6e-3	Corner	-1e-2	-3e-3
4	1e-3	6e-3	Corner	9e-3	-3e-3

In this model, rectangles will be added to the boundary of the rectangle to define the location of the positions of the contacts and the floating contacts. Using the menu bar, select Draw > Specify Objects > Rectangle. In the Draw > Specify Objects > Rectangle edit window, individually create each of the rectangles shown in Table 4.17 by selecting the window, entering the data, and then clicking OK. The final geometry with all four added rectangles is shown in Figure 4.114.

Click the Zoom Extents button. Select Draw > Create Composite Object. Select all of the rectangles. Verify that the Keep interior boundaries check box is checked. see Figure 4.113. Click OK. Figure 4.114 shows the composite object.

NOTE The contact rectangles are added to the boundary of the first rectangle so that the contacts and the floating contacts are precisely defined.

2D Hall Effect Subdomain Settings

Using the menu bar, select Physics > Subdomain Settings. Select subdomains 1, 3, 4, and 5 in the Subdomain selection window. Enter t_Si in the d (Thickness) edit window.

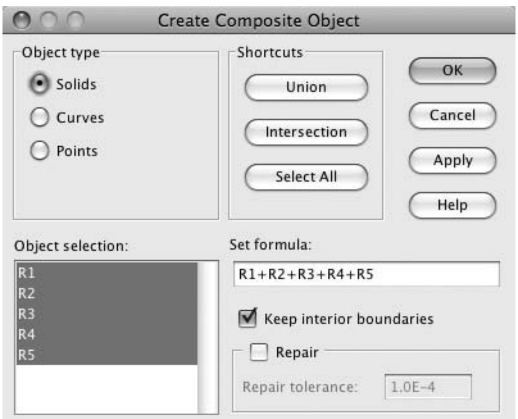


FIGURE 4.113 2D Hall_Effect_2 model, Create Composite Object window

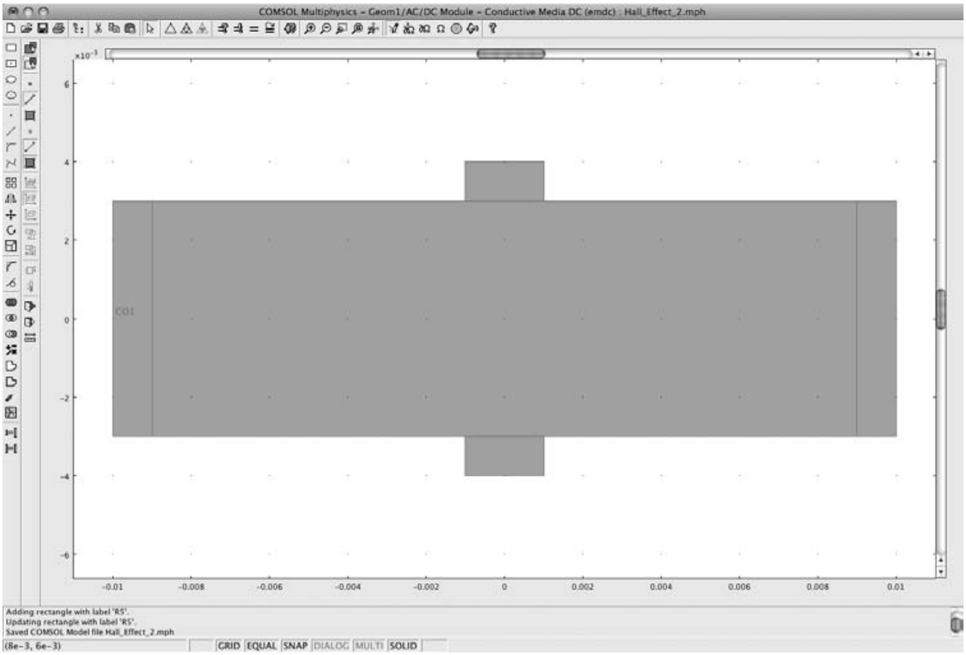


FIGURE 4.114 2D Hall_Effect_2 model geometry with added rectangles

Verify that "Conductivity" is selected in the Conductivity relation pull-down list. Enter sigma0 in the Electric conductivity window. See Figure 4.115.

Select subdomain 2 in the Subdomain selection window. Enter: t_Si in the d (Thickness) edit window. Verify that "Conductivity" is selected in the Conductivity relation pull-down list. Click in the Electric conductivity edit window. Select "Anisotropic-full" from the Conductivity type pull-down list. Enter the matrix elements as shown in Table 4.18; see Figures 4.116 and 4.117. Click OK.

NOTE These matrix elements define the anisotropic coupling of the magnetic field and the current flowing in the silicon sample.

Table 4.18 Matrix Elements Edit Window

Matrix Element Number	Value
11	s11
12	s12
21	s21
22	s22

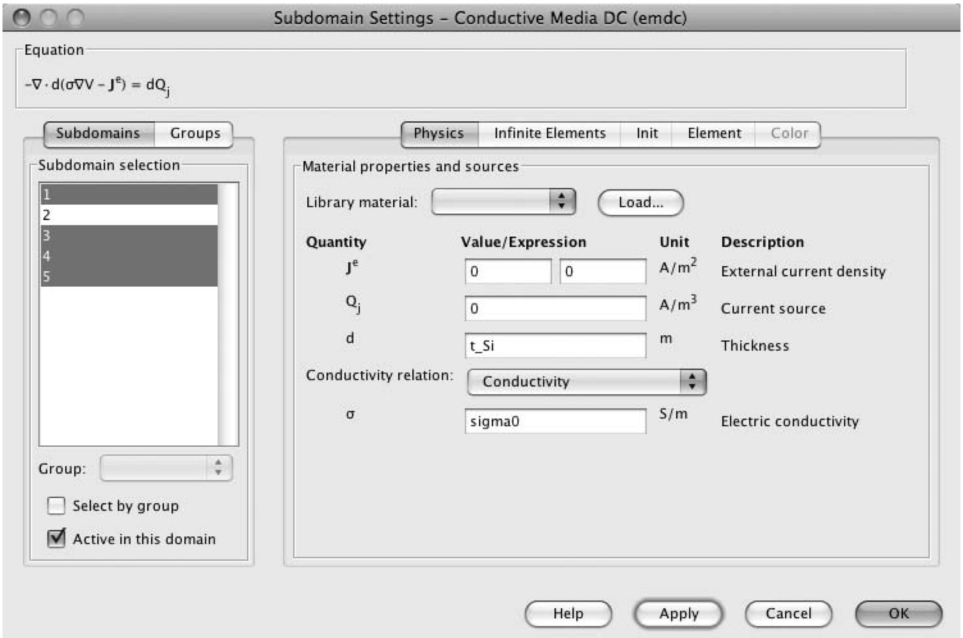


FIGURE 4.115 2D Hall_Effect_2 model geometry, Subdomain Settings (1, 3, 4, 5)

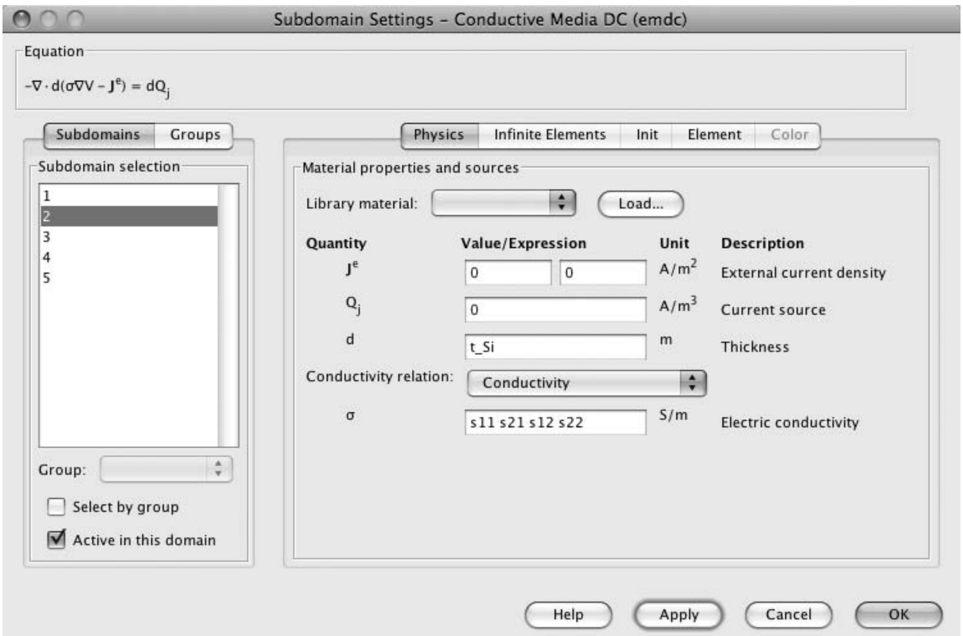


FIGURE 4.116 2D Hall_Effect_2 model geometry, Subdomain Settings (2)

Table 4.19 Boundary Settings

Boundary Number	Condition	Group Index	Source Current/Potential	Figure
1	Ground	—	—	4.118
2, 3, 5–7, 10, 13–16, 18, 19	Electric insulation	—	—	4.119
8	Floating potential	2	0	4.120
12	Floating potential	1	0	4.121
20	Electric potential	—	V0	4.122

2D Hall Effect Boundary Settings

Using the menu bar, select Physics > Boundary Settings. Enter the boundary settings as shown in Table 4.19. See Figures 4.118, 4.119, 4.120, 4.121, and 4.122.

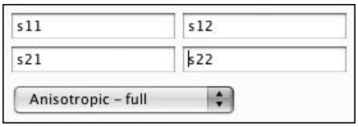


FIGURE 4.117 2D Hall_Effect_2 model conductivity matrix elements

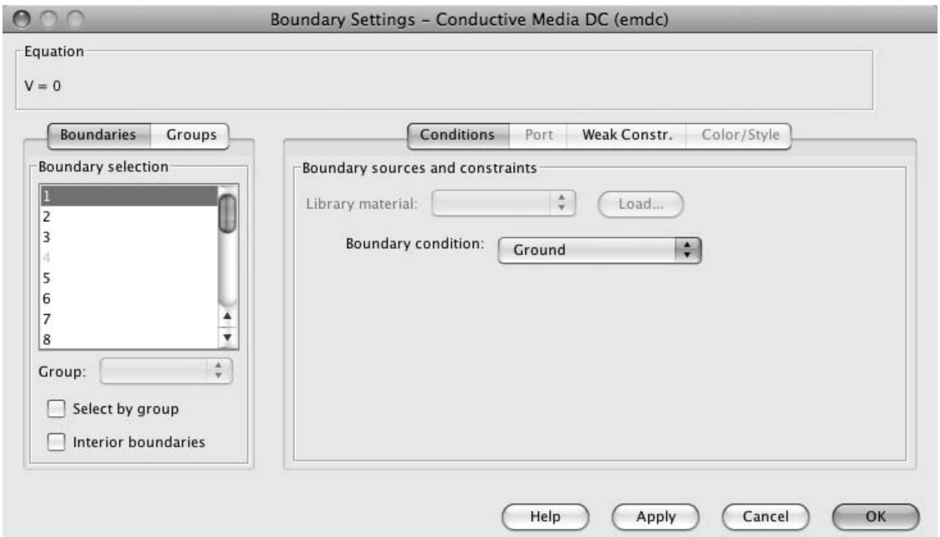


FIGURE 4.118 2D Hall_Effect_2 model Boundary Settings (1)

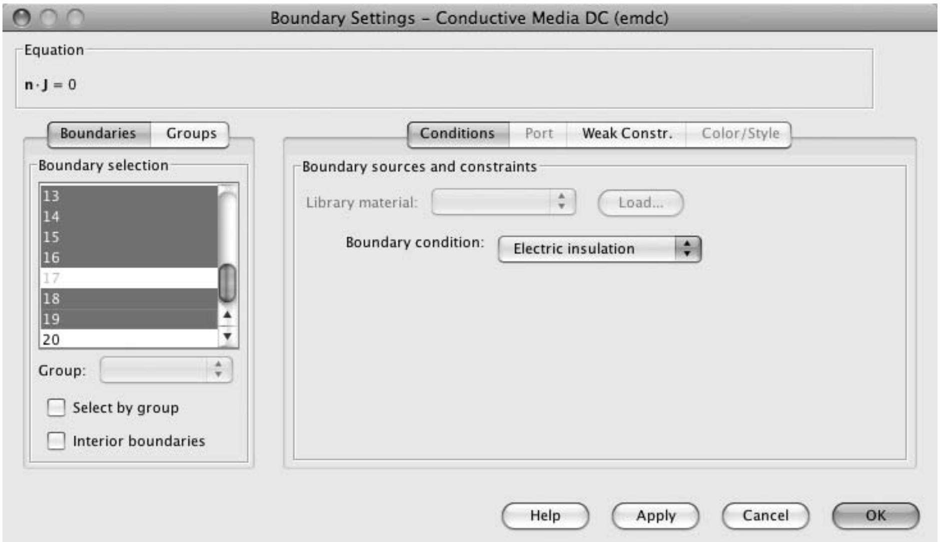


FIGURE 4.119 2D Hall_Effect_2 model Boundary Settings (2, 3, 5–7, 10, 13–16, 18, 19)

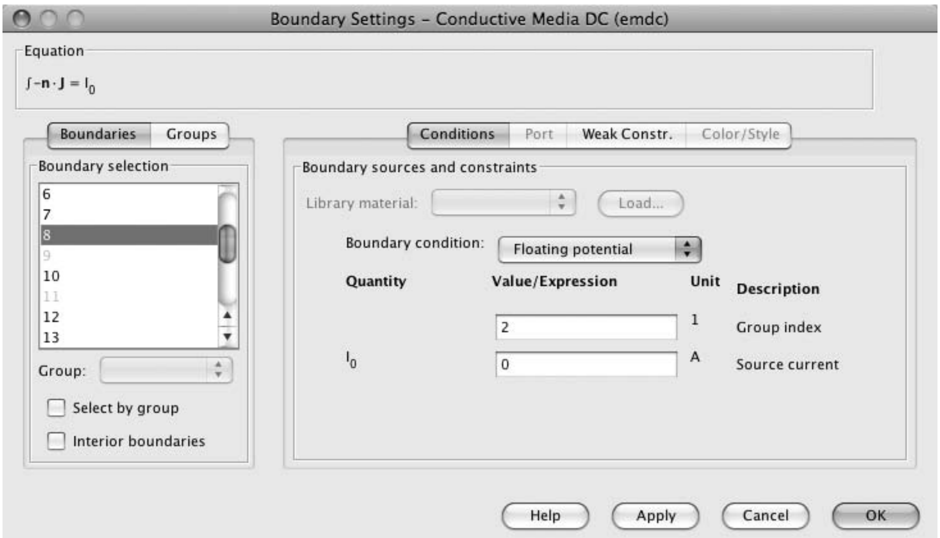


FIGURE 4.120 2D Hall_Effect_2 model Boundary Settings (8)

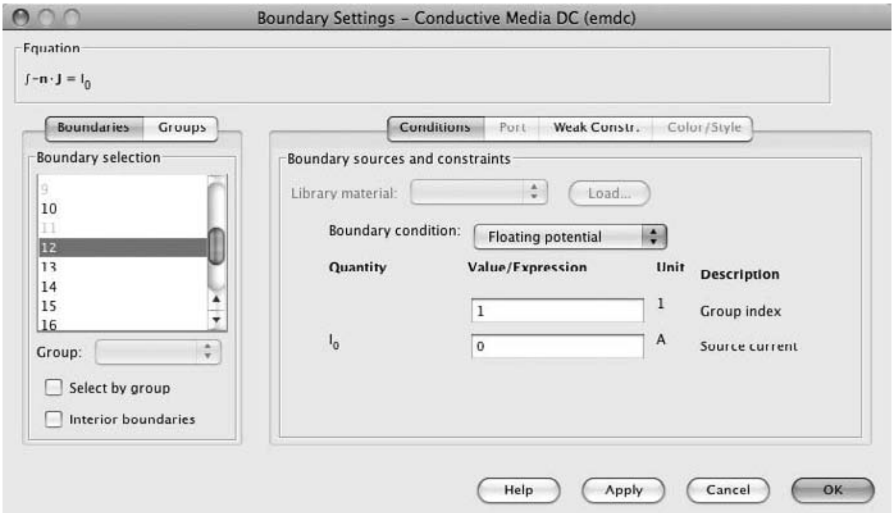


FIGURE 4.121 2D Hall_Effect_2 model Boundary Settings (12)

NOTE The addition of the group index designation decouples the two floating contacts from each other.

Select the Weak Constr. tab. Verify that the Use weak constraints check box is checked. See Figure 4.123.

Click OK. The final configuration of the boundary settings is shown in Figure 4.124.

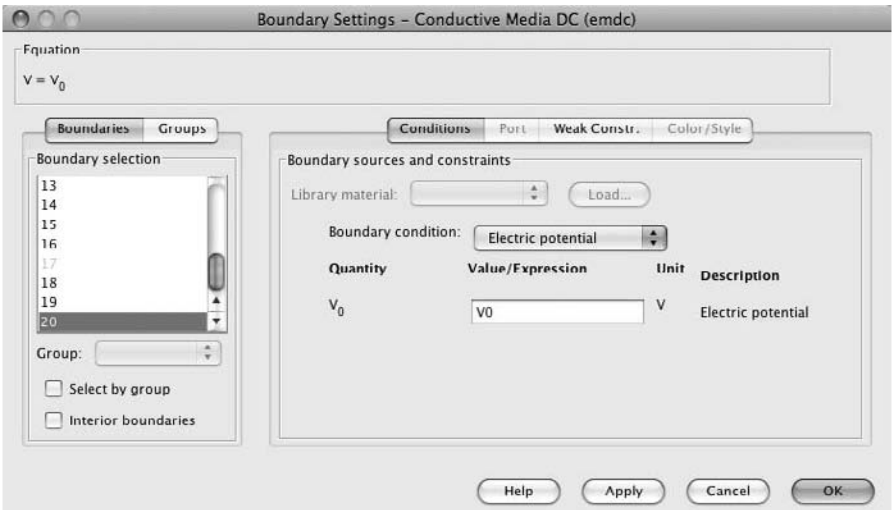


FIGURE 4.122 2D Hall_Effect_2 model Boundary Settings (20)

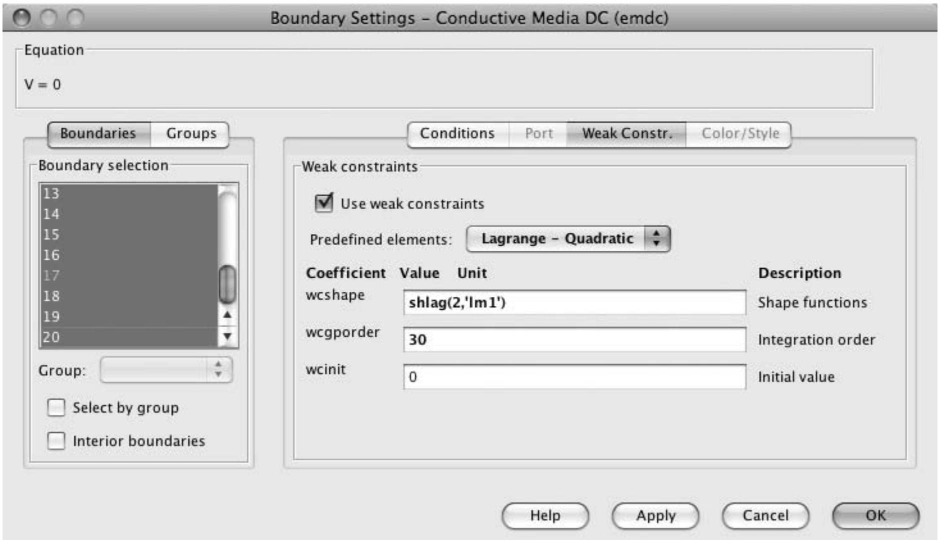


FIGURE 4.123 2D Hall_Effect_2 model Boundary Settings, Weak Constr. page

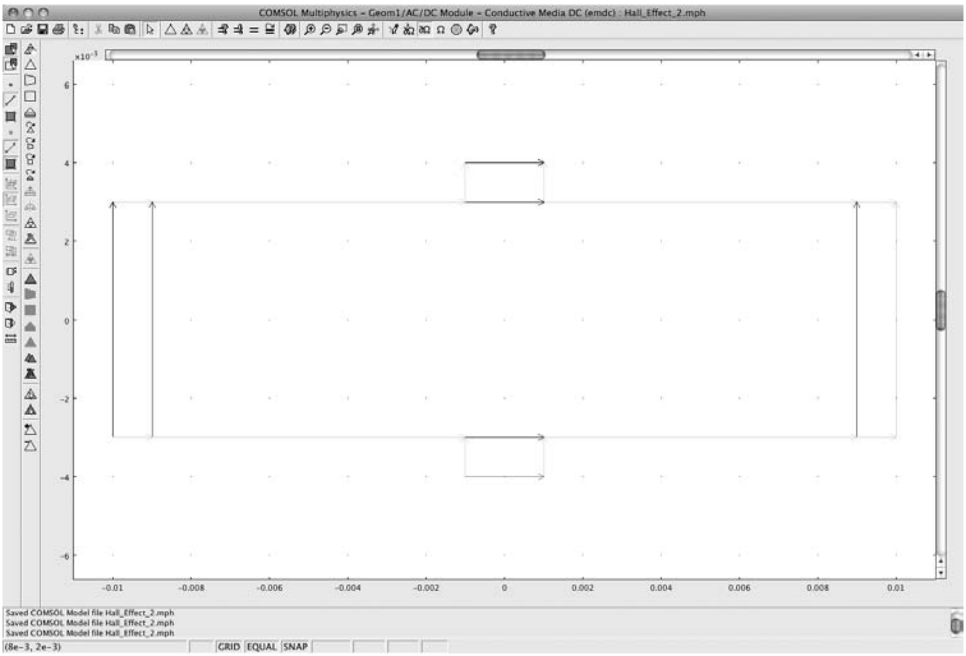


FIGURE 4.124 2D Hall_Effect_2 model boundary settings, final configuration

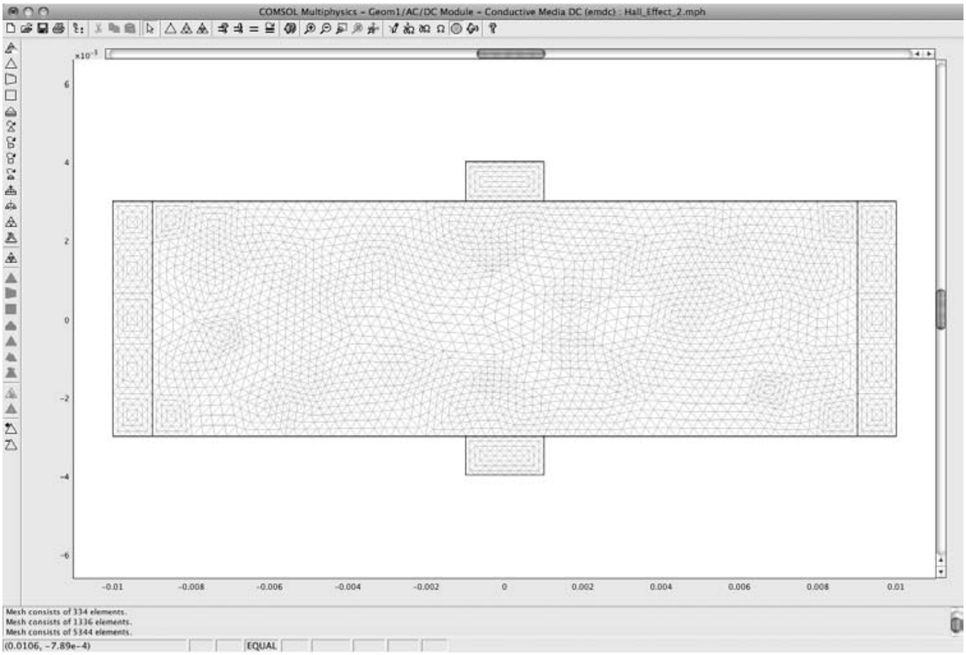


FIGURE 4.125 2D Hall_Effect_2 model mesh

2D Hall Effect Mesh Generation

Using the menu bar, select Mesh > Initialize Mesh. Click the Refine Mesh button twice. See Figure 4.125.

Solving the First Variation on the 2D Hall Effect Model

Using the menu bar, select Solve > Solver Parameters. In the Solver selection window, select "Parametric." In the Parameter name edit window, enter Bz. In the Parameter values edit window, enter 0:0.1:2.0. See Figure 4.126. Click OK.

NOTE The Parametric Solver is chosen, in this case, so that the modeler can solve the Hall_Effect_2 Model quasi-statically. This allows the modeler to see solutions over a wide range of magnetic field values.

Using the menu bar, select Solve > Solve Problem.

Postprocessing

The default plot is a 2D surface plot of the voltage distribution at the highest value of the magnetic field (Bz = 2 Tesla). See Figure 4.127.

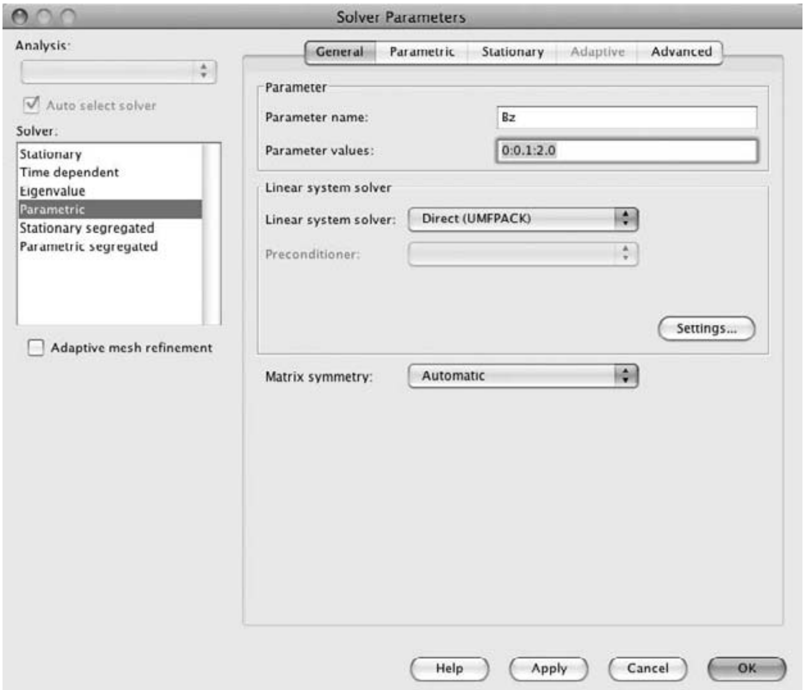


FIGURE 4.126 2D Hall_Effect_2 model Solver Parameters window

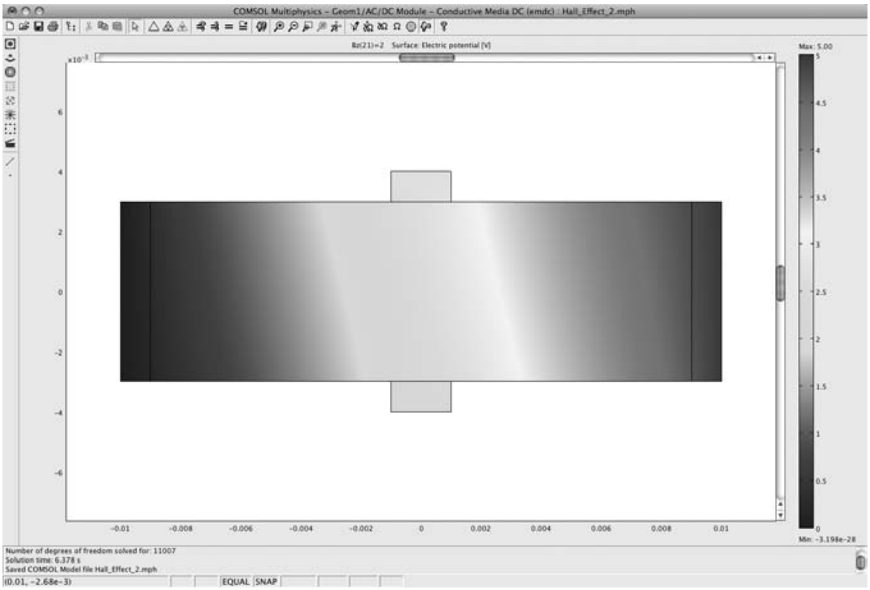


FIGURE 4.127 2D Hall_Effect_2 model default surface voltage distribution plot

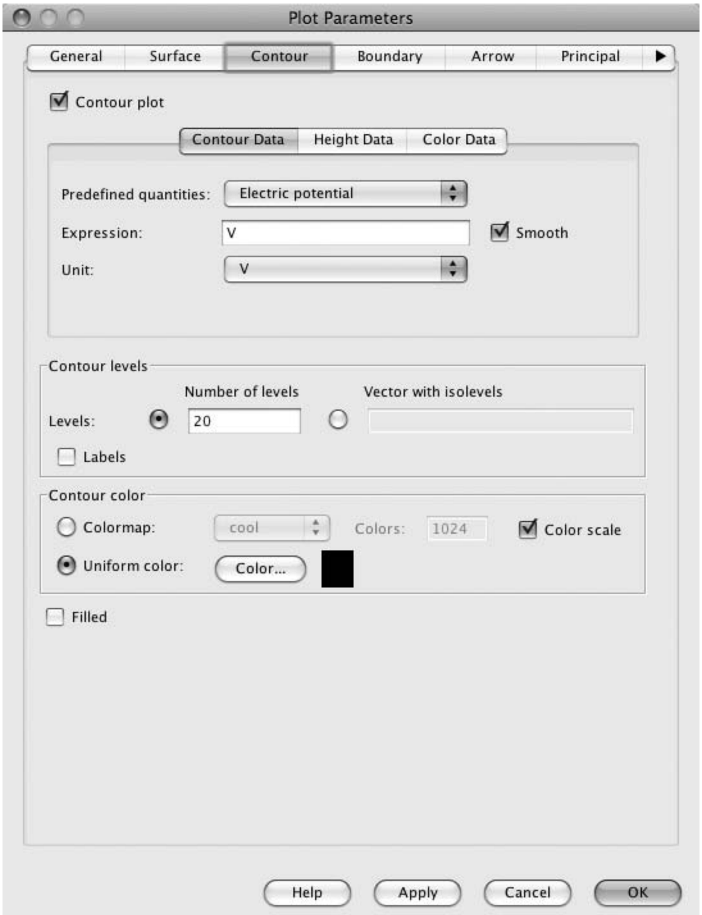


FIGURE 4.128 2D Hall_Effect_2 model Plot Parameters, Contour Data page

More detailed information is displayed by adding contour lines. Using the menu bar, select Postprocessing > Plot Parameters. Click the Contour tab. Place a check mark in the Contour plot check box. Click the Uniform color radio button. Click the Color select button, and select "Black." See Figure 4.128.

Click OK. The Hall effect voltage (V_H) can be seen as the voltage difference between the top electrode and the bottom electrode, as shown in Figure 4.129.

The exact voltage difference at any point in the model can be determined by creating a cross-section plot. Using the menu bar, select Postprocessing > Cross-Section Plot Parameters. Select the 2T solution in the Solutions to use window. See Figure 4.130.

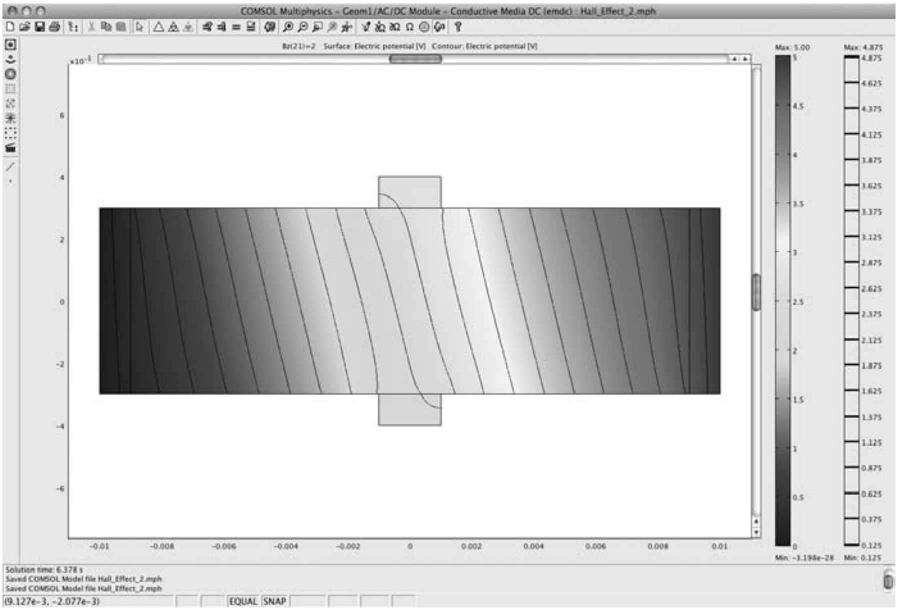


FIGURE 4.129 2D Hall_Effect_2 model surface voltage distribution plot (2T), with contour lines

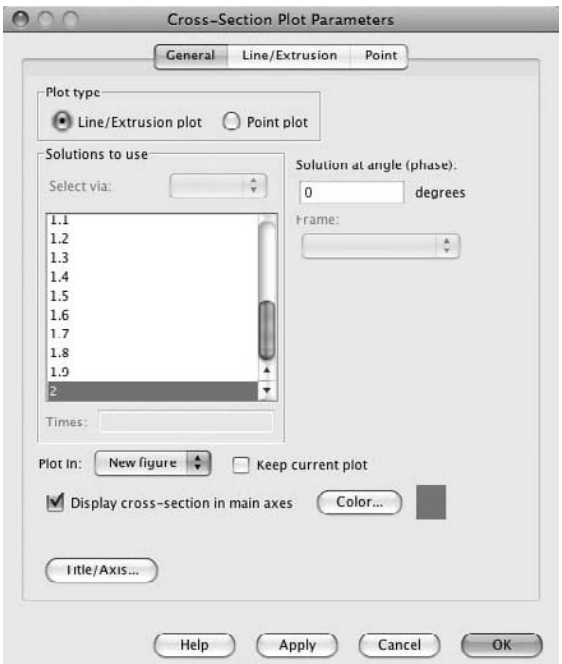


FIGURE 4.130 2D Hall_Effect_2 model Cross-Section Plot Parameters, General page

Table 4.20 Cross-Section Line Data Edit Window

Line Data	Value
x0	0e-3
x1	0e-3
y0	-4e-3
y1	4e-3

Click the Line/Extrusion tab. Enter the coordinates shown in Table 4.20 on the Cross-Section Plot Parameters page. See Figure 4.131.

Click OK. Figure 4.132 shows the voltage difference (V_H) between the electrode (top) and the modeled Si sample (bottom), for the line $x = 0$. In this case, $V_H = 0.350$ volts ($V_{high} - V_{low} = 0.350$ V).

Postprocessing Animation

This solution to the 2D Hall_Effect_2 model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 4.133). Click the

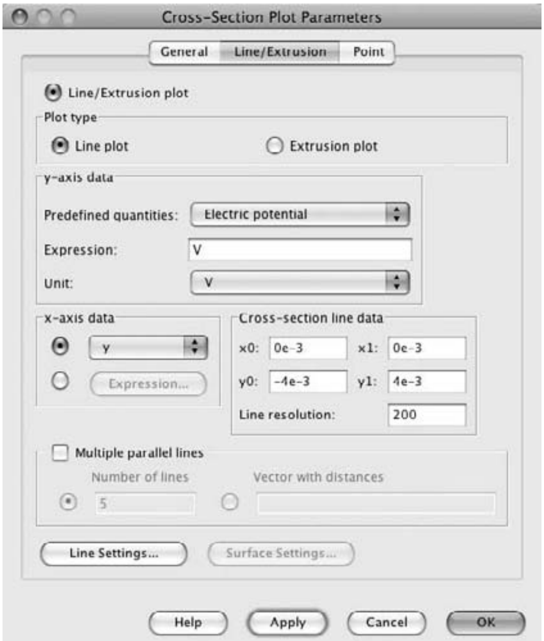


FIGURE 4.131 2D Hall_Effect_2 model Cross-Section Plot Parameters, Line/Extrusion page

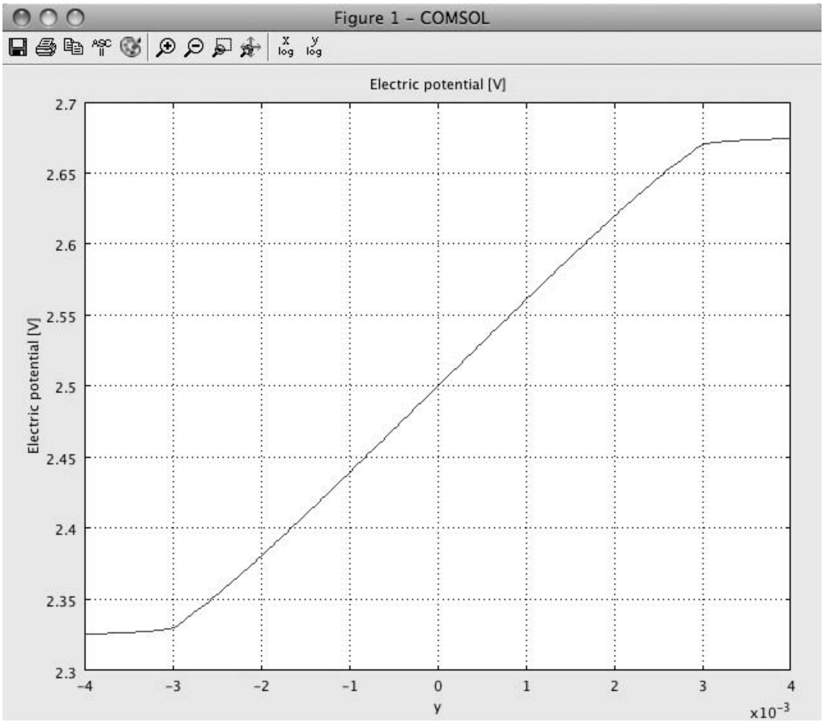


FIGURE 4.132 2D Hall_Effect_2 model plot V_H

Start Animation button. Save this 2D Hall effect model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4_HE_2.avi that was supplied with this book.

Second Variation on the 2D Hall Effect Model

This Hall effect model demonstrates the effect of having holes as the carrier in this electronic material (a p-type semiconductor). A second, lower contact has been added to allow the use of an external balancing circuit. A change is made in the value of the conductivity, because holes are less mobile than electrons in Si.

NOTE Semiconductors have two types of carriers: electrons (n-type) and holes (p-type). In a purified semiconductor, such as silicon (Si) or germanium (Ge), both carriers are thermally activated and exist in equal numbers. That native conduction mode is called the intrinsic conduction mode.

To fabricate electronic device structures, foreign atoms (As or P for n-type and Al or B for p-type) are added to the host lattice (Si). The carriers are more easily thermally activated from the foreign atoms (dopant atoms) at room temperature. This non-native

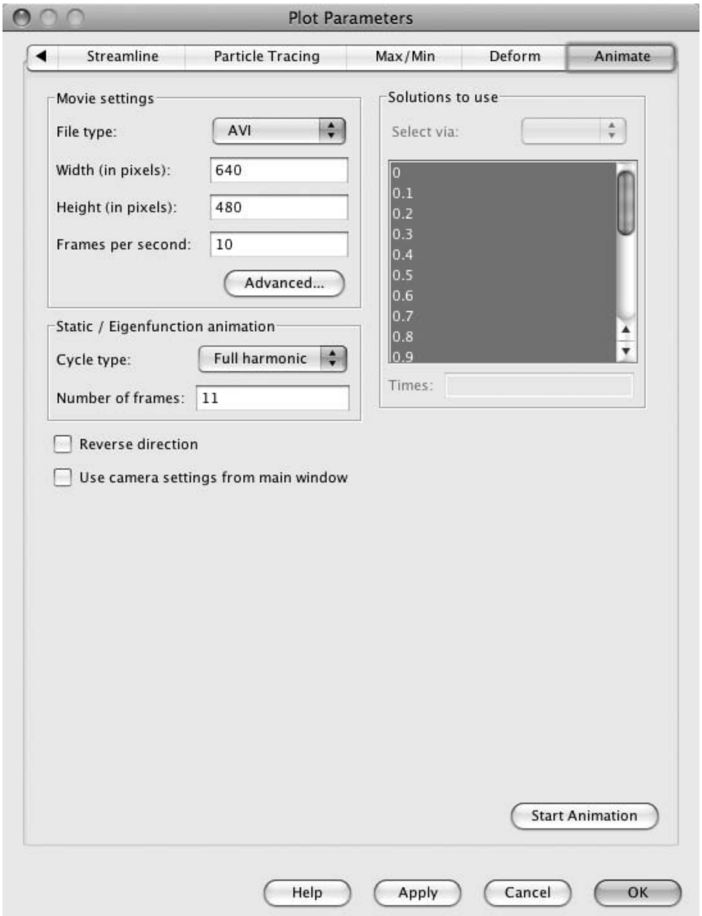


FIGURE 4.133 2D Hall_Effect_2 model animation Plot Parameters window

conduction mode is called the extrinsic conduction mode. In the extrinsic mode, the carriers activated from the dopant atoms and the small number of carriers activated intrinsically become the majority carriers (e.g., electrons). The second carrier (holes in this example) becomes the minority carrier. The electron and hole carrier densities are related by the mass action law:[18]

$$np = n_i^2 \tag{4.10}$$

where n_i = intrinsic carrier density
n = electron carrier density
p = hole carrier density

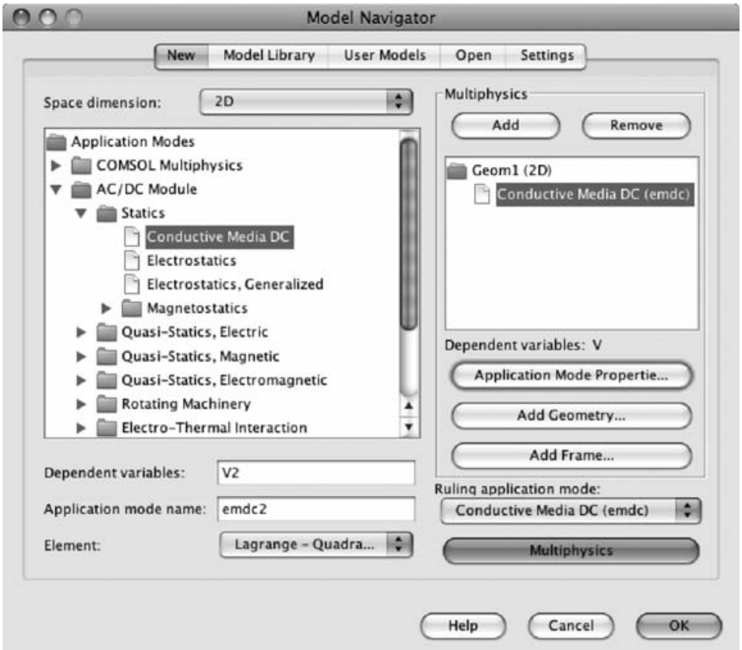


FIGURE 4.134 Multiphysics Model Navigator window

To start building the Hall_Effect_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select AC/DC Module > Statics > Conductive Media DC. Click the Multiphysics button, and then click the Add button. See Figure 4.134.

Click the Application Mode Properties button. Select "On" from the Weak constraints pull-down list. Select "Non-ideal" from the Constraint type pull-down list. See Figure 4.135. Click OK.

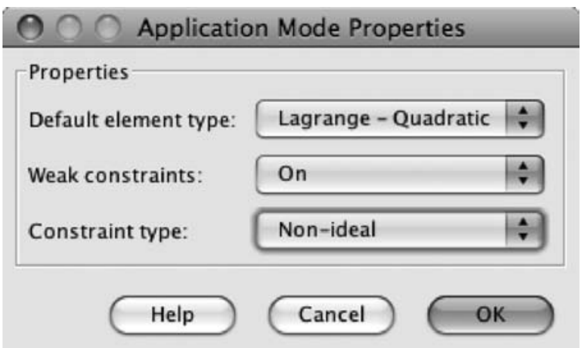


FIGURE 4.135 Application Mode Properties window

Table 4.21 Constants Edit Window

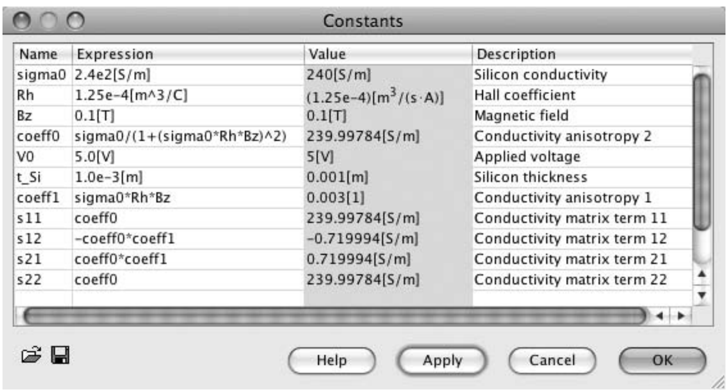
Name	Expression	Description
sigma0	2.4e2[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	Hall coefficient
Bz	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	Conductivity anisotropy 2
V0	5.0[V]	Applied voltage
t_Si	1.0e-3[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	Conductivity anisotropy 1
s11	coeff0	Conductivity matrix term 11
s12	-coeff0*coeff1	Conductivity matrix term 12
s21	coeff0*coeff1	Conductivity matrix term 21
s22	coeff0	Conductivity matrix term 22

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.21; also see Figure 4.136. Click OK.

2D Hall Effect Geometry

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 1.8e-2, and a height of 6e-3. Select "Base: Corner" and set x equal to -9e-3, and y equal to -3e-3 in the Rectangle edit window. See Figure 4.137.



Name	Expression	Value	Description
sigma0	2.4e2[S/m]	240[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	(1.25e-4)[m^3/(s·A)]	Hall coefficient
Bz	0.1[T]	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	239.99784[S/m]	Conductivity anisotropy 2
V0	5.0[V]	5[V]	Applied voltage
t_Si	1.0e-3[m]	0.001[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	0.003[1]	Conductivity anisotropy 1
s11	coeff0	239.99784[S/m]	Conductivity matrix term 11
s12	-coeff0*coeff1	-0.719994[S/m]	Conductivity matrix term 12
s21	coeff0*coeff1	0.719994[S/m]	Conductivity matrix term 21
s22	coeff0	239.99784[S/m]	Conductivity matrix term 22

FIGURE 4.136 2D Hall_Effect_3 model Constants edit window

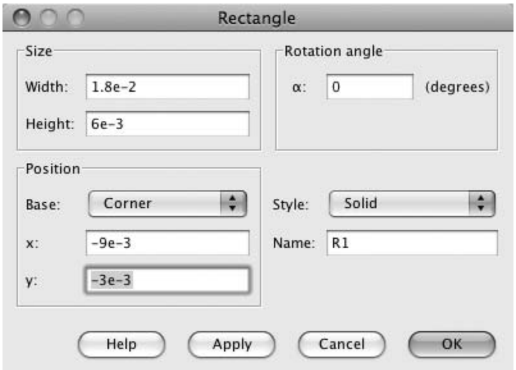


FIGURE 4.137 2D Hall_Effect_3 model Rectangle edit window

Click OK, and then click the Zoom Extents button. See Figure 4.138.

In this model, rectangles will be added to the boundary of the rectangle to define the location of the positions of the contacts and the floating contacts. Using the menu bar, select Draw > Specify Objects > Rectangle. In the Draw > Specify Objects > Rectangle edit window, individually create each of the rectangles shown in Table 4.22 by selecting the window, entering the data, and then clicking OK.

Click the Zoom Extents button. Select Draw > Create Composite Object. Select all of the rectangles. Verify that the Keep interior boundaries check box is checked. See Figure 4.139.

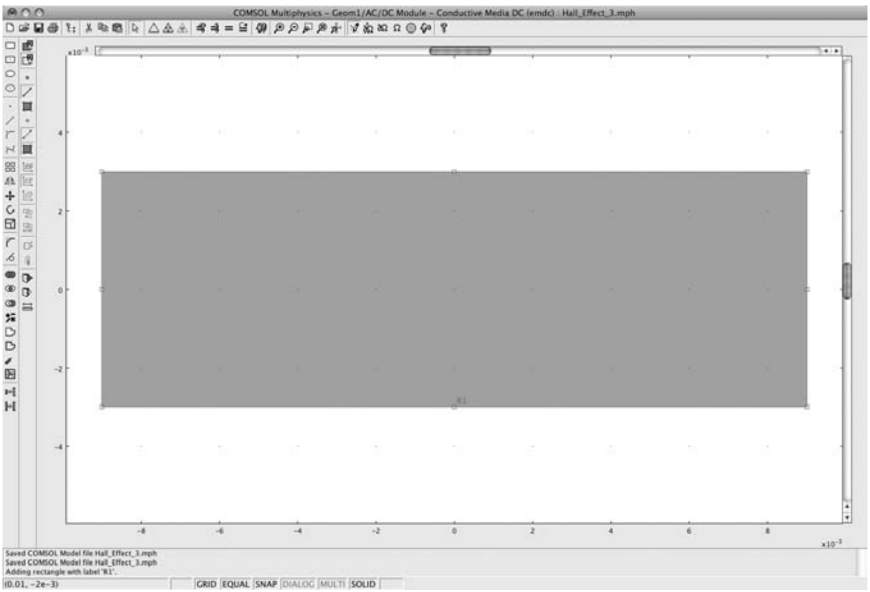


FIGURE 4.138 2D Hall_Effect_3 model rectangle geometry

Table 4.22 Rectangle Edit Window

Rectangle Number	Width	Height	Base	*x* Location	*y* Location
1	2e-3	1e-3	Corner	-1e-3	3e-3
2	2e-3	1e-3	Corner	-4e-3	-4e-3
3	2e-3	1e-3	Corner	2e-3	-4e-3
4	1e-3	6e-3	Corner	-1e-2	-3e-3
5	1e-3	6e-3	Corner	9e-3	-3e-3

Click OK. Figure 4.140 shows the composite object.

NOTE One contact rectangle at the top of the Hall effect model, two contact rectangles at the bottom, and two contact rectangles on the ends are added to the boundary of the first rectangle so that the contacts and the floating contacts are precisely defined. The three contacts (top and bottom) are a typical experimental configuration to allow the measuring instrument to balance the circuit and offset any unintended error voltages.

2D Hall Effect Subdomain Settings

Using the menu bar, select Physics > Subdomain Settings. Select subdomains 1, 3, 4, 5, and 6 in the Subdomain selection window. Enter t_Si in the d (Thickness) edit window. Verify that "Conductivity" is selected in the Conductivity relation pull-down list. Enter sigma0 in the Electric conductivity window. See Figure 4.141.

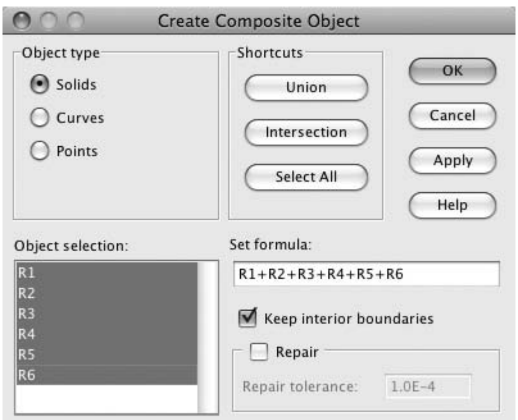


FIGURE 4.139 2D Hall_Effect_3 model, Create Composite Object window

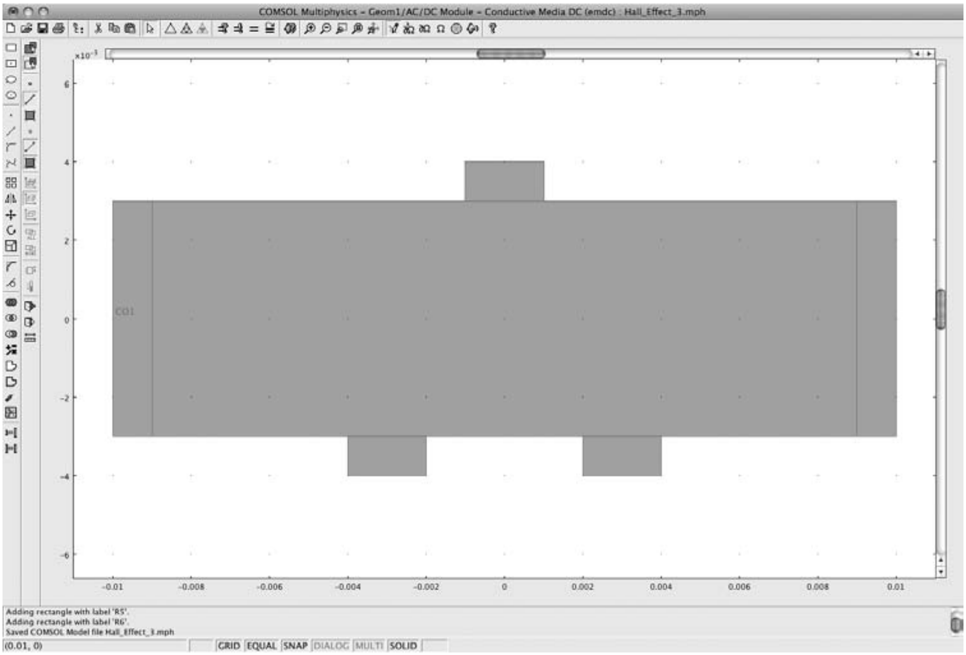


FIGURE 4.140 2D Hall_Effect_3 model geometry with added rectangles

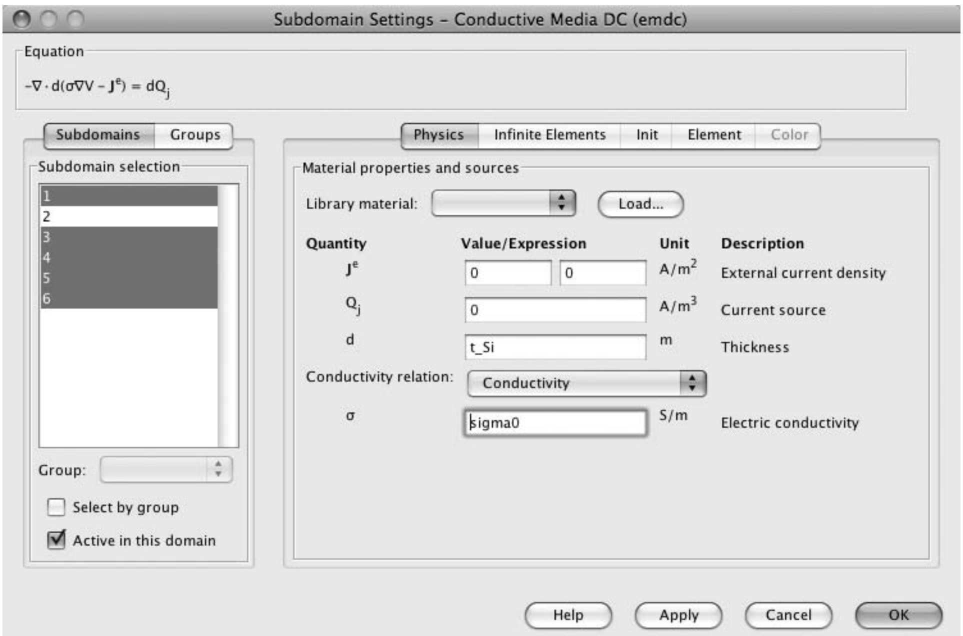


FIGURE 4.141 2D Hall_Effect_3 model geometry, Subdomain Settings (1, 3, 4, 5, 6)

Table 4.23 Matrix Elements Edit Window

Matrix Element Number	Value
11	s11
12	s12
21	s21
22	s22

NOTE These matrix elements define the anisotropic coupling of the magnetic field and the current flowing in the silicon sample.

Select subdomain 2 in the Subdomain selection window. Enter t_Si in the d (Thickness) edit window. Verify that "Conductivity" is selected in the Conductivity relation pull-down list. Click in the Electric conductivity edit window. Select "Anisotropic-full" from the Conductivity type pull-down list. Enter the matrix elements as shown in Table 4.23; see Figures 4.142 and 4.143. Click OK.

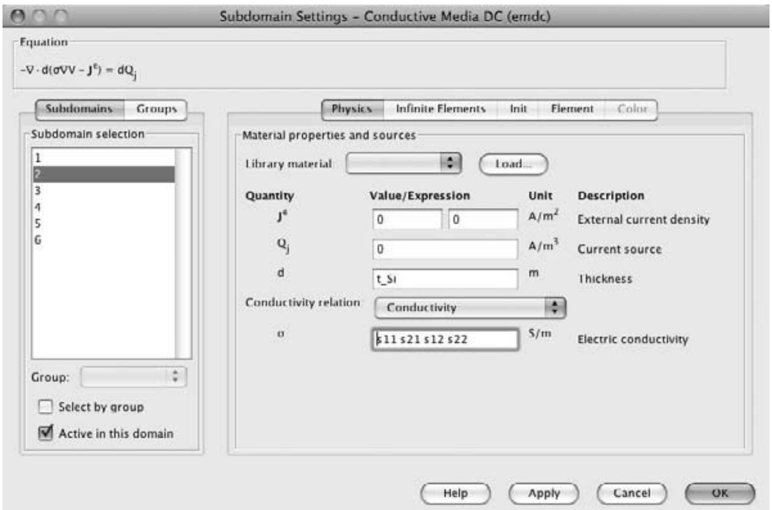


FIGURE 4.142 2D Hall_Effect_3 model geometry, Subdomain Settings (2)

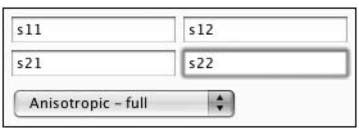


FIGURE 4.143 2D Hall_Effect_3 model conductivity matrix elements

Table 4.24 Boundary Settings

Boundary Number	Condition	Group Index	Source Current/Potential	Figure
1	Ground	—	—	4.144
2, 3, 5–7, 10–12, 15–17, 20, 21 23, 24	Electric insulation	—	—	4.145
8	Floating potential	1	0	4.146
14	Floating potential	2	0	4.147
18	Floating potential	3	0	4.148
25	Electric potential	—	V0	4.149

NOTE The addition of the group index designation decouples the three floating contacts from one another.

2D Hall Effect Boundary Settings

Using the menu bar, select Physics > Boundary Settings. Enter the boundary settings as shown in Table 4.24. See Figures 4.144, 4.145, 4.146, 4.147, 4.148, and 4.149.

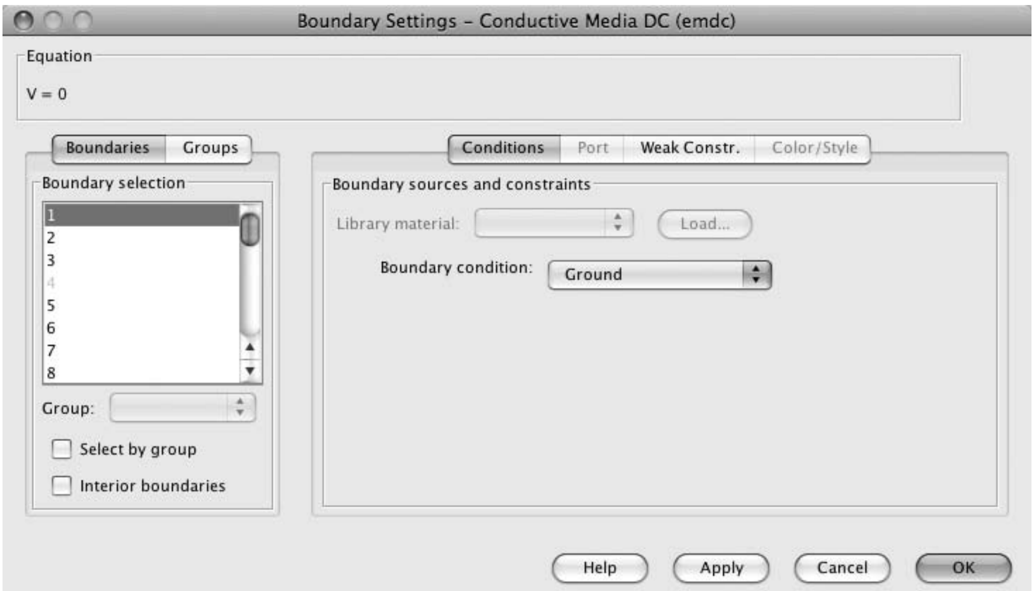


FIGURE 4.144 2D Hall_Effect_3 model Boundary Settings (1)

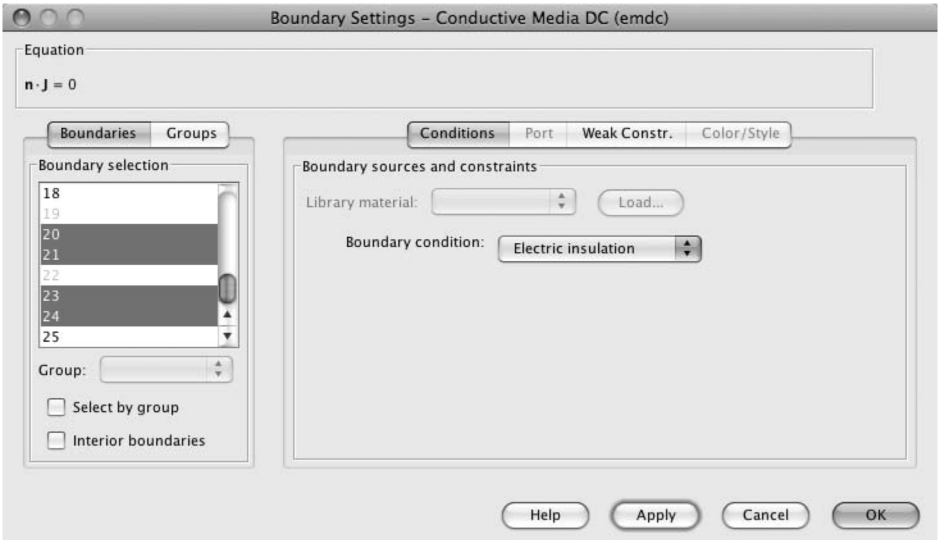


FIGURE 4.145 2D Hall_Effect_3 model Boundary Settings (2, 3, 5–7, 10–12, 15–17, 20, 21, 23, 24)

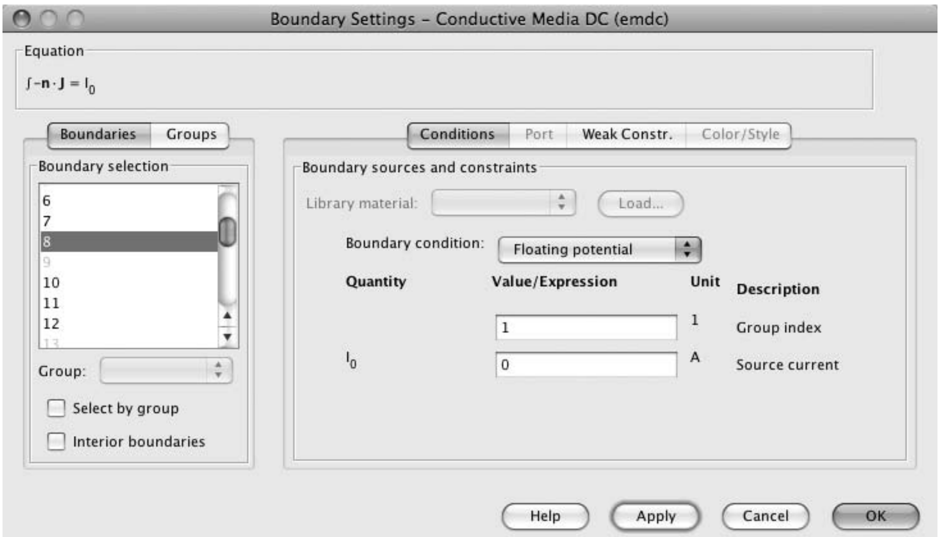


FIGURE 4.146 2D Hall_Effect_3 model Boundary Settings (8)

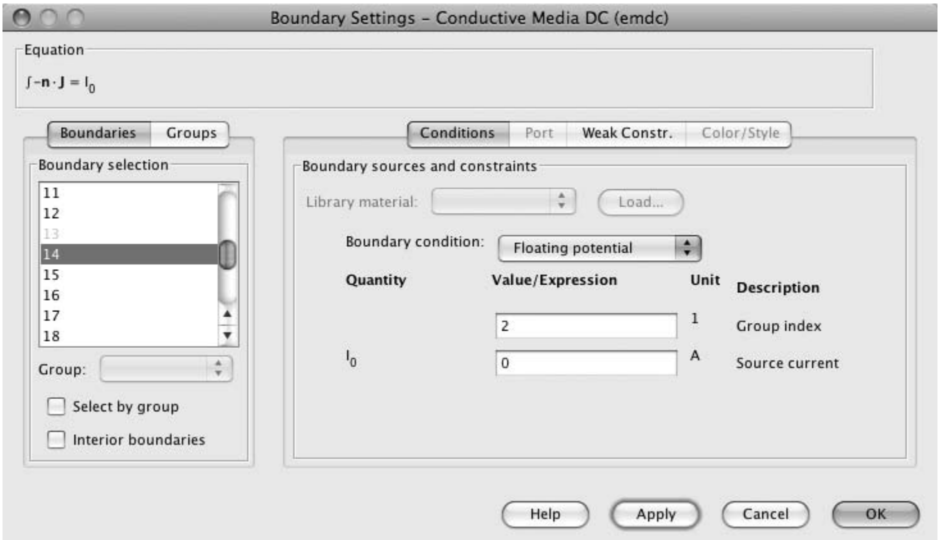


FIGURE 4.147 2D Hall_Effect_3 model Boundary Settings (14)

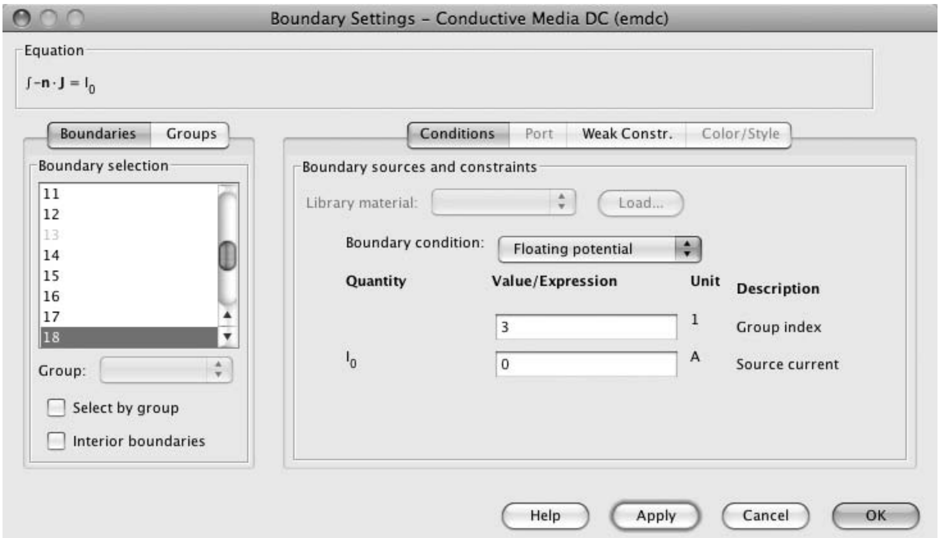


FIGURE 4.148 2D Hall_Effect_3 model Boundary Settings (18)

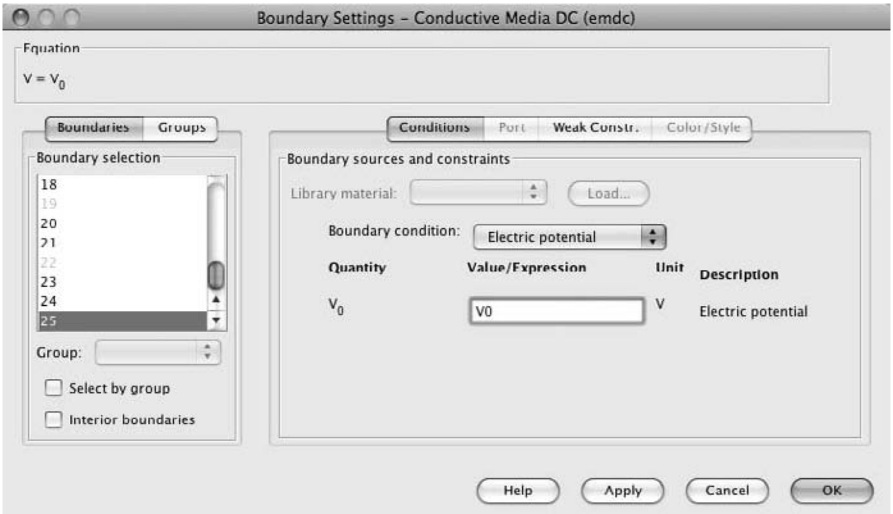


FIGURE 4.149 2D Hall_Effect_3 model Boundary Settings (25)

Click the Weak Constr. tab. Verify that the Use weak constraints check box is checked. See Figure 4.150.

Click OK. The final configuration of the boundary settings is shown in Figure 4.151.

2D Hall Effect Mesh Generation

Using the menu bar, select Mesh > Initialize Mesh. Click the Refine Mesh button twice. See Figure 4.152.

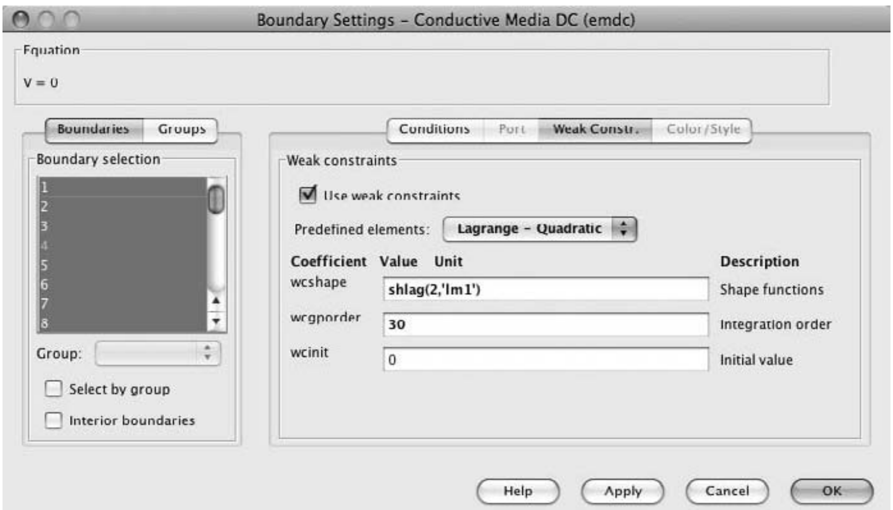


FIGURE 4.150 2D Hall_Effect_3 model Boundary Settings, Weak Constr. page

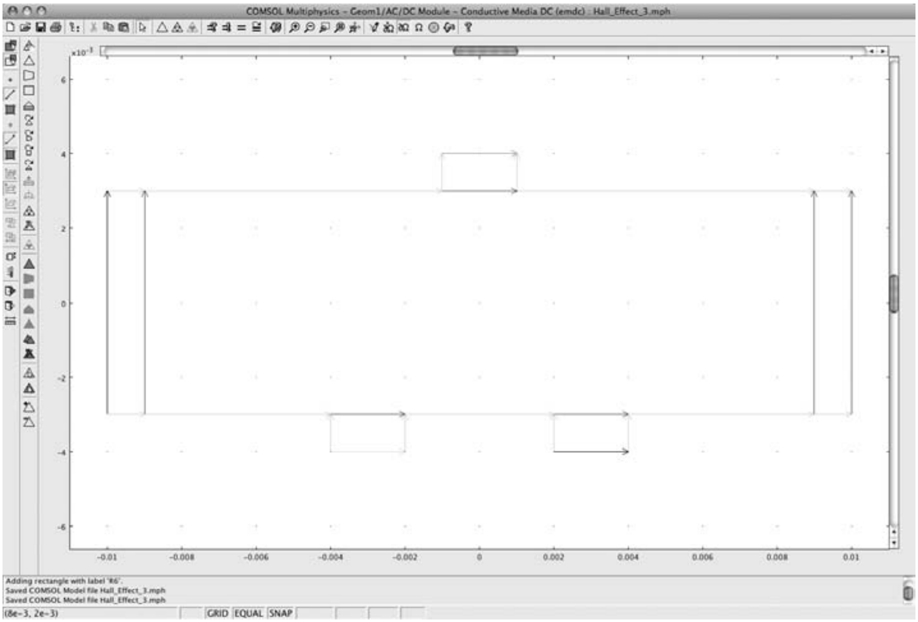


FIGURE 4.151 2D Hall_Effect_3 model boundary settings, final configuration

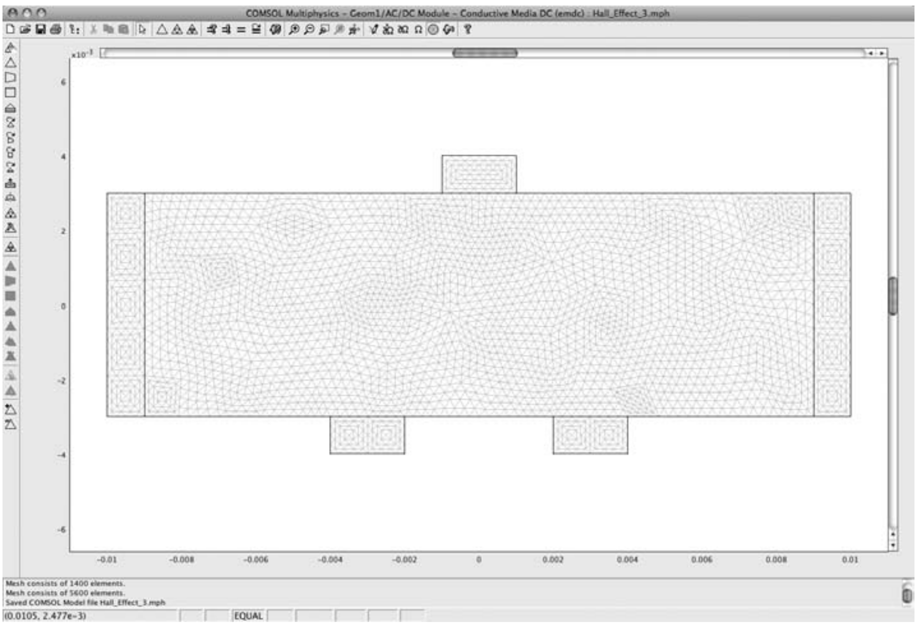


FIGURE 4.152 2D Hall_Effect_3 model mesh

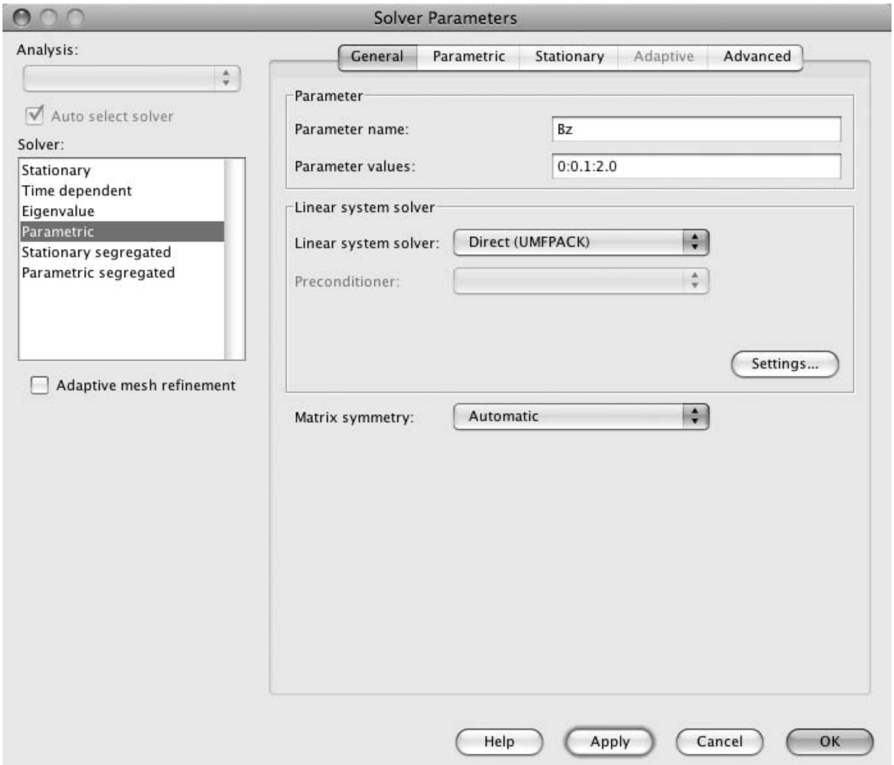


FIGURE 4.153 2D Hall_Effect_3 model Solver Parameters window

Solving the Second Variation on the 2D Hall Effect Model

Using the menu bar, select Solve > Solver Parameters. In the Solver selection window, select "Parametric." In the Parameter name edit window, enter Bz. In the Parameter values edit window, enter 0:0.1:2.0. See Figure 4.153. Click OK.

NOTE The Parametric Solver is chosen, in this case, so that the modeler can solve the Hall_Effect_3 Model quasi-statically. This allows the modeler to see solutions over a wide range of magnetic field values.

Using the menu bar, select Solve > Solve Problem.

Postprocessing

The default plot is a 2D surface plot of the voltage distribution at the highest value of the magnetic field (Bz = 2 Tesla). See Figure 4.154.

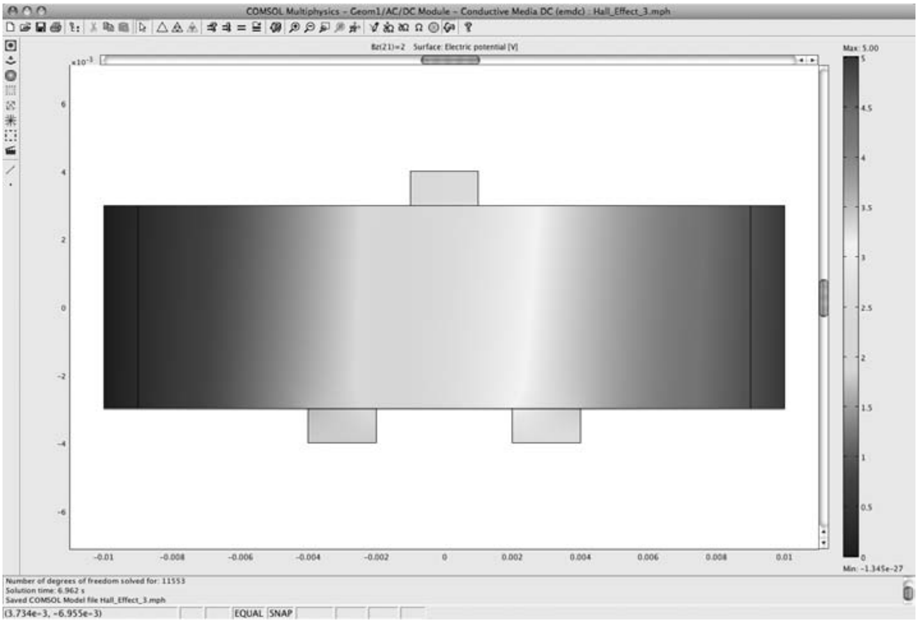


FIGURE 4.154 2D Hall_Effect_3 model default surface voltage distribution plot

More detailed information is displayed by adding contour lines. Using the menu bar, select Postprocessing > Plot Parameters. Click the Contour tab. Place a check mark in the Contour plot check box. Click the Uniform color radio button. Click the Color select button, and select “Black.” See Figure 4.155.

Click OK. See Figure 4.156.

The exact voltage difference at any point in the model can be determined by creating a cross-section plot. Using the menu bar, select Postprocessing > Cross-Section Plot Parameters. Select the 2T solution in the Solutions to use window. See Figure 4.157.

Click the Line/Extrusion tab. Enter the coordinates shown in Table 4.25 on the Cross-Section Plot Parameters page.

Table 4.25 Cross-Section Line Data Edit Window

Line Data	Value
x0	0e-3
x1	0e-3
y0	-3e-3
y1	4e-3

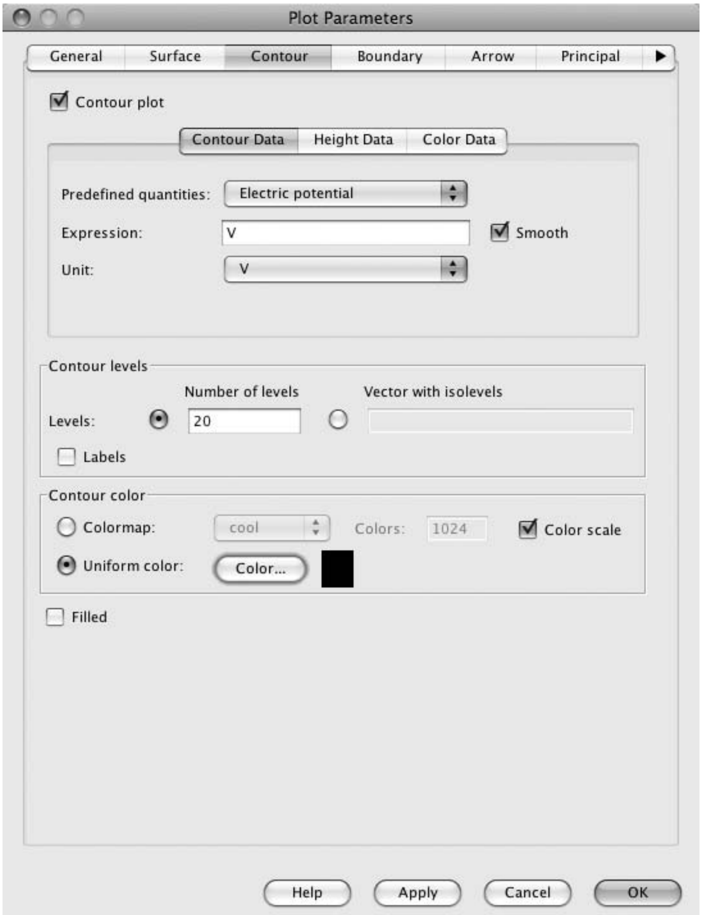


FIGURE 4.155 2D Hall_Effect_3 model Plot Parameters, Contour Data page

Select "y" on the x-axis data pull-down list. See Figure 4.158.

Click OK. Figure 4.159 shows the voltage difference (V_H) between the electrode (top) and the modeled Si sample (bottom), for the line $x = 0$. In this case, $V_H = 0.085$ volts ($V_{high} - V_{low} = 0.085$ V).

Postprocessing Animation

This solution to the 2D Hall_Effect_3 model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 4.160). Click the Start Animation button. Save this 2D Hall effect model animation by

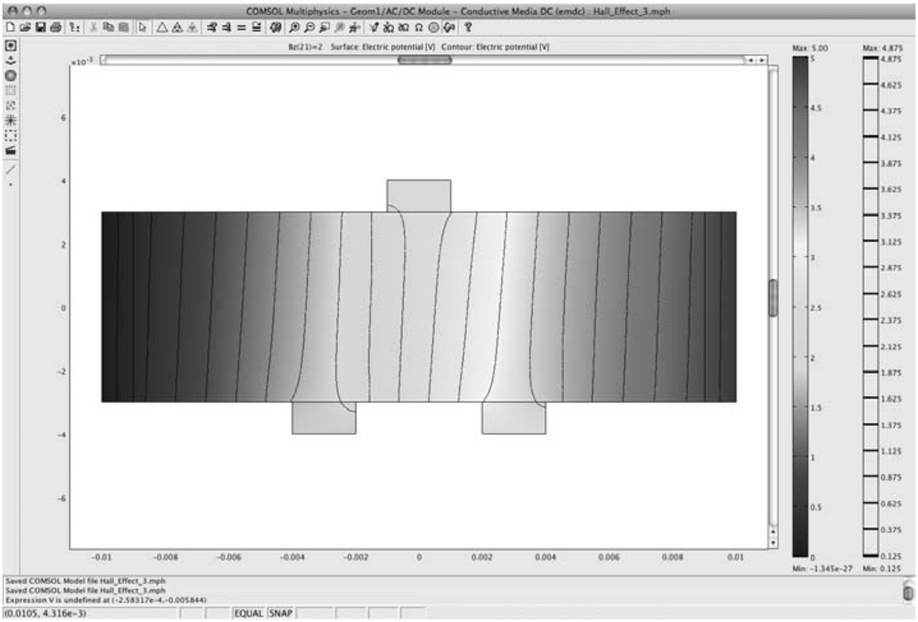


FIGURE 4.156 2D Hall_Effect_3 model surface voltage distribution plot (2T), with contour lines

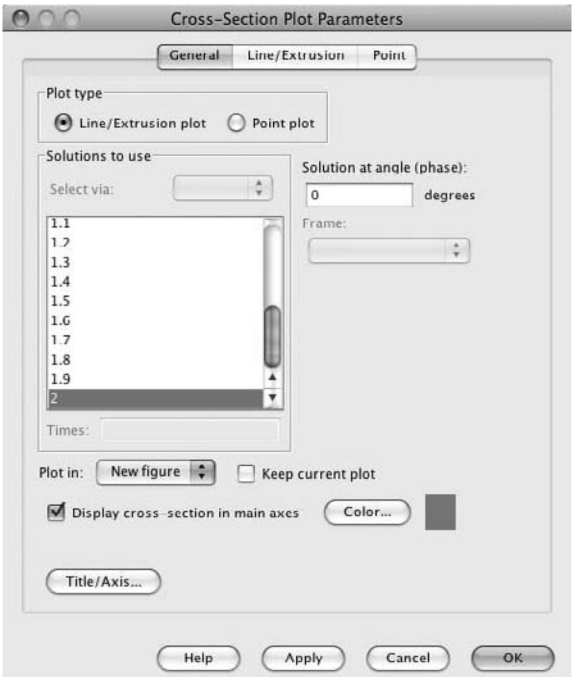


FIGURE 4.157 2D Hall_Effect_3 model Cross-Section Plot Parameters, General page

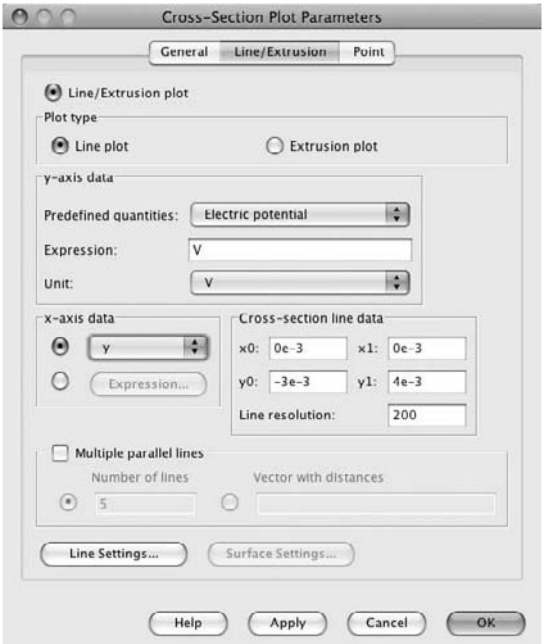


FIGURE 4.158 2D Hall_Effect_3 model Cross-Section Plot Parameters, Line/Extrusion page

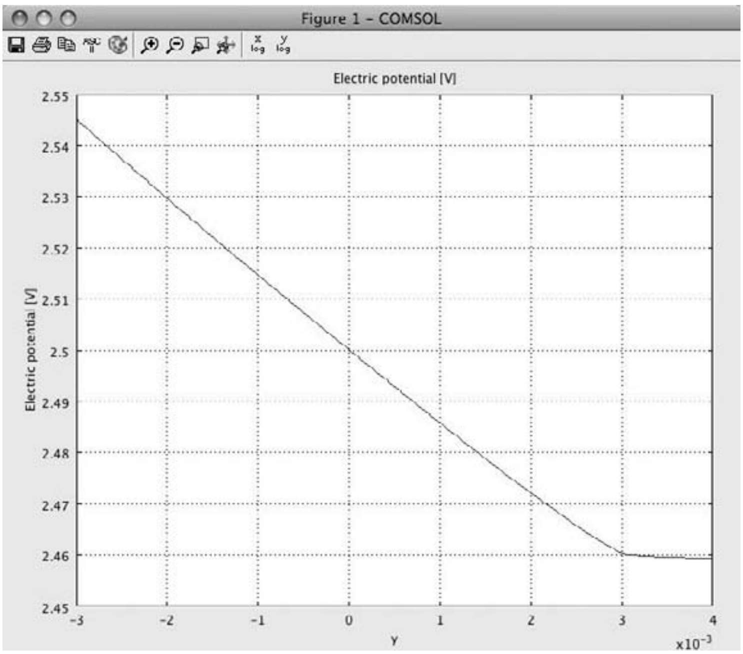


FIGURE 4.159 2D Hall_Effect_3 model plot V_H

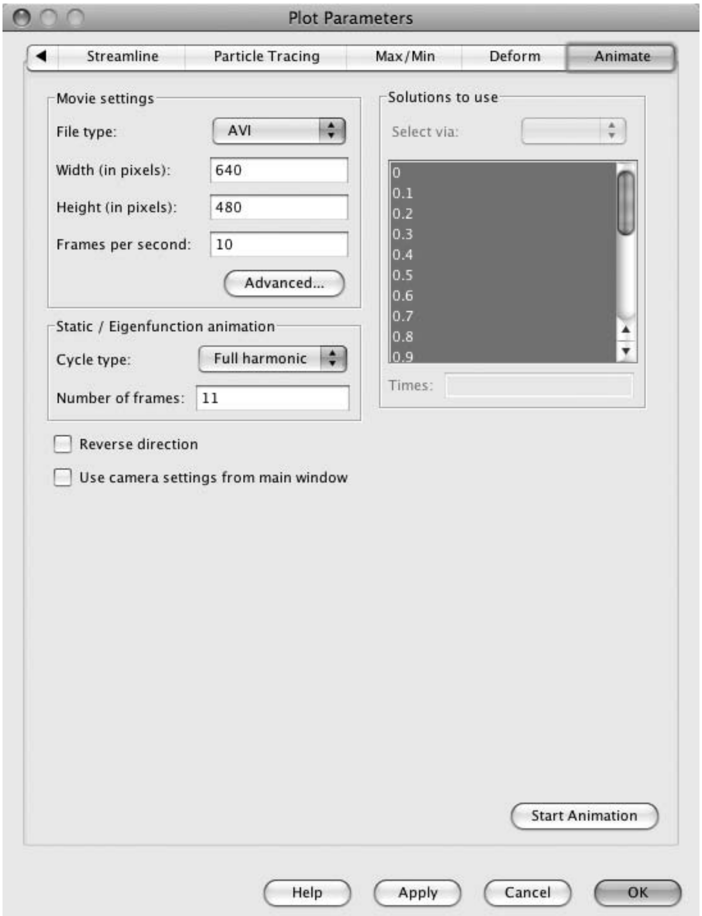


FIGURE 4.160 2D Hall_Effect_3 model animation Plot Parameters window

clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4_HE_3.avi that was supplied with this book.

2D Hall Effect Models: Summary and Conclusions

The models presented in this section of Chapter 4 have introduced the following new concepts: two-dimensional modeling (2D), the Hall effect, AC/DC Module, Conductive Media DC, weak constraints, floating contacts, anisotropic conductivity, semiconductor dual-carrier types, and imbalance-offset geometry. The 2D Hall effect model is a powerful tool that can be used to model Hall effect magnetic sensors for sensing fluid flow, rotating or linear motion, proximity, current, pressure, and orientation. A comparison of the calculated results for the three Hall effect models is shown in Table 4.26.

Table 4.26 Hall Effect Modeling Results Summary

Model	Floating Contacts	Carrier Type	V_H (V)	(V_H △%)
HE_1	Point defined	n-type (electron)	0.340	—
HE_2	Dual rectangle defined	n-type (electron)	0.350	~+3
HE_3	Triple rectangle defined	p-type (hole)	−0.085	~−75

The differences between the calculations for the tested models for the n-type carrier are in the range of a few percentage points. It can clearly be seen that the p-type silicon is only one-fourth as sensitive as the n-type material. That reduction in sensitivity is attributable to the low hole mobility. It is left to the modeler to explore other differences between the models by variation of the parameters, as suggested in the exercises at the end of this chapter.

References

1. http://en.wikipedia.org/wiki/Electrochemical
2. http://en.wikipedia.org/wiki/Electropolishing
3. http://en.wikipedia.org/wiki/William_Gilbert
4. http://en.wikipedia.org/wiki/Charles-Augustin_de_Coulomb
5. http://en.wikipedia.org/wiki/Joseph_Priestley
6. http://en.wikipedia.org/wiki/Georg_Ohm
7. http://en.wikipedia.org/wiki/Michael_Faraday
8. "Deformed Meshes" in *COMSOL Multiphysics Modeling Guide, Version 3.4,* 392–405.
9. http://en.wikipedia.org/wiki/Ohm%27s_law
10. C. Kittel, *Introduction to Solid State Physics* New York: John Wiley & Sons, 1986), 206–214.
11. S. M. Sze, *Semiconductor Devices, Physics and Technology* (New York: John Wiley & Sons), 1985, 34–40.
12. J. M. Ziman, *Principles of the Theory of Solids* (Cambridge, UK: Cambridge University Press), 1969, 185.
13. http://en.wikipedia.org/wiki/Hall_Effect
14. http://en.wikipedia.org/wiki/Edwin_Hall
15. http://en.wikipedia.org/wiki/Lorentz_Force

16. R. A. Smith, *Semiconductors* (Cambridge, UK: Cambridge University Press, 1968), 100–107.
17. COMSOL Multiphysics Software Models Database, Hall Plate with Floating Contacts.
18. S. M. Sze, *Semiconductor Devices, Physics and Technology* (New York: John Wiley & Sons), 1985, 16–20.

Exercises

1. Build, mesh, and solve the COMSOL 2D electrochemical polishing model problem presented in this chapter.
2. Build, mesh, and solve the first variation of the 2D electrochemical polishing model problem presented in this chapter.
3. Build, mesh, and solve the second variation of the 2D electrochemical polishing model problem presented in this chapter.
4. Build, mesh, and solve the Hall effect model presented in this chapter.
5. Build, mesh, and solve the first variation of the Hall effect model presented in this chapter.
6. Build, mesh, and solve the second variation of the Hall effect model presented in this chapter.
7. Explore other variations of the arguments in the COMSOL 2D electrochemical polishing models.
8. Explore other variations of the arguments in the Hall effect models.
9. Explore how an increase in the run time modifies the behavior of the COMSOL 2D electrochemical polishing model.
10. Explore how changes in the sample geometry affect the behavior of the Hall effect model.

5 2D Axisymmetric Modeling

In This Chapter

■ 2D Axisymmetric Guidelines for New COMSOL® Multiphysics® Modelers

2D Axisymmetric Modeling Considerations

2D axisymmetric modeling can be less difficult than 1D modeling and is about the same level of difficulty as 2D modeling. Specifically, 2D axisymmetric modeling has fewer implicit assumptions than 1D modeling. The 2D axisymmetric model requires the modeler to think in terms of cylindrical coordinates and rotational symmetry. Such models can be challenging to build, depending on the underlying physics involved. The least difficult aspect of 2D axisymmetric model building arises from the fact that the geometry is still relatively simple. (In a 2D model, the modeler has only a single plane as the modeling space.) However, the physics and the rotational nature of the geometry in a 2D axisymmetric model can range from relatively easy to extremely complex.

NOTE COMSOL® Multiphysics® software has two 2D modeling modes: 2D (beginning-level through advanced-level 2D modeling) and 2D axisymmetric (advanced-level 2D modeling). In keeping with the introductory focus of the material in this text, both of the model types—that is, the 2D model introduced in Chapter 4, and the 2D axisymmetric model introduced in this chapter—along with the associated physics and the related methodology for use of the models, are introduced in this book. Significantly more advanced 2D modeling techniques exist than those presented here in Chapters 4 and 5. Examples of some of those more difficult techniques are reserved for introduction in later chapters (6 and 7). For further expansion of your 2D modeling horizons, refer to the COMSOL Manuals, the COMSOL website, and the general COMSOL Multiphysics software-related research literature.

The 2D axisymmetric model implicitly assumes, in compliance with the laws of physics, that the energy flow, the materials properties, the environment, and any other conditions and variables that are of interest are homogeneous, isotropic, or constant, unless otherwise specified, throughout the entire domain of interest, both within the model and, through the boundary conditions, in the environs of the model. Bearing that in mind, the modeler needs to ensure that all of the modeling conditions and associated parameters (default settings) in each new model created have been properly considered, defined, or set to the appropriate value(s).

The modeler also needs to seriously consider the steps that will be required to establish the correct postprocessing and visualization settings so as to extract the desired information from the modeling solution. The default parameter settings on any given model will probably not present exactly the information that the modeler needs or desires, although it may come close to meeting the modeler's demands. It is the responsibility of the modeler to determine exactly which of the myriad of postprocessing and visualization choices available in the COMSOL Multiphysics software to employ.

NOTE As mentioned previously, it is always preferable for the modeler to be able to accurately anticipate the expected behavior (results) of the model and the presentation of its results. Do not assume that the default values that are initially present when the model is first created will suit the needs of the new model. Always verify that the values employed in the model are the correct values needed for that model. Calculated solution values that significantly deviate from the expected values or from comparison values measured in experimentally derived realistic models are probably indicative of one or more modeling errors either in the original model design, in the earlier model analysis, or in the understanding of the underlying physics, or they may simply be due to human error.

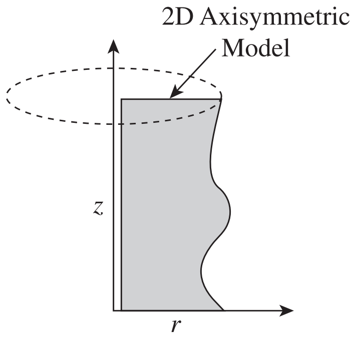


FIGURE 5.1 2D axisymmetric coordinate system

2D Axisymmetric Coordinate System

In 2D axisymmetric models, there are two geometrical coordinates: space (r) and space (z). See Figure 5.1.

In the steady-state solution to any 2D axisymmetric model, parameters can vary only as a function of the radial position in space (r) and the axial position space (z) coordinates. Such a model represents the parametric condition of the model in a time-independent mode (quasi-static). In a transient solution model, parameters can vary both by position in space (r) and space (z) and in time (t). The transient solution model is essentially a sequential collection of steady-state (quasi-static) solutions. The space coordinates (r) and (z) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values ($r_{min} <= r <= r_{max}$) and ($z_{min} <= z <= z_{max}$). The time coordinate (t) represents the range of values ($t_{min} <= t <= t_{max}$) from the beginning of the observation period (t_{min}) to the end of the observation period (t_{max}).

To assist the reader in achieving a broader exposure to the applicability of the physics discussed in this chapter and to demonstrate the power of the basic COMSOL 2D axisymmetric modeling techniques, the examples in this chapter demonstrate heat transfer modeling techniques from two substantially different approaches. Heat transfer is an extremely important design consideration. It is one of the most widely needed and applied technologies employed in applied engineering and physics. Most modern products or processes require an understanding of heat transfer either during development or during the use of the product or process (e.g., automobiles, plate glass fabrication, plastic extrusion, plastic products, houses, ice cream).

NOTE Heat transfer concerns have existed since the beginning of prehistory. The science of thermodynamics, and consequently the present understanding of heat transfer, started with the work of Nicolas Leonard Sadi Carnot, as published in his 1824 paper titled "Reflections on the Motive Power of Fire."[1] The first use of the term "thermodynamics" is attributed to William Thomson (Lord Kelvin).[2] Subsequent contributions to the understanding of heat, heat transfer, and thermodynamics in general were made by James Prescott Joule,[3] Ludwig Boltzmann,[4] James Clerk Maxwell,[5] Max Planck,[6] and numerous others. The physical understanding and engineering use of thermodynamics play a very important role in the technological aspects of machine and process design in modern applied science, engineering, and medicine.

The first example presented in this chapter, on cylinder conduction, explores the 2D axisymmetric steady-state modeling of heat transfer and temperature profiling for a thermally conductive material, implemented through use of the COMSOL Heat Transfer Module. In the first variation on the cylinder conduction model, a model is built using the basic COMSOL Multiphysics software. The calculated modeling results are then compared. The second variation on the cylinder conduction model explores the use of heat transfer modeling for low-pressure gas/vacuum environments.

The second 2D axisymmetric modeling example in this chapter, titled Thermos_Container, explores the modeling of heat loss for thermally insulated containers.

NOTE Insulated containers can be found applied in many different applications in modern society. Examples include Thermos containers, water heaters, and refrigerated liquid containers (for fuels, liquified gases, heat exchangers, and so on).

Heat Conduction Theory

Heat conduction is a naturally occurring process that is readily observed in many aspects of modern life (e.g., refrigerators, freezers, microwave ovens, thermal ovens, engines). The heat transfer process allows both linear and rotational work to be done in the generation of electricity and the movement of vehicles. The initial understanding of transient heat transfer was developed by Newton[7] and started with Newton's law of cooling:[8]

$$\frac{dQ}{dt} = h^*A^*(T_S - T_E) \tag{5.1}$$

where $\frac{dQ}{dt}$ is the incremental energy lost in joules per unit time (J/s)

A is the energy transmission surface area (m^2)

h is the heat transfer coefficient [W/(m^2*K)]

T_S is the surface temperature of the object losing heat (K)

T_E is the temperature of the environment gaining heat (K)

Subsequent work by Jean Baptiste Joseph Fourier,[9] based on Newton's law of cooling, developed the law for steady-state heat conduction (known as Fourier's law[10]). Fourier's law is expressed here in differential form:

$$q = -k\nabla T \tag{5.2}$$

where q is the heat flux in watts per square meter (W/m^2)
k is the thermal conductivity of the material [W/(m*K)]
∇T is the temperature gradient (K/m)

2D Axisymmetric Heat Conduction Modeling

The following numerical solution model (cylinder conduction) was originally developed by COMSOL as a tutorial model based on an example from the NAFEMS collection.[11] It was developed for distribution with the Multiphysics software as a COMSOL Multiphysics General Heat Transfer Application Mode Model in the Heat Transfer Module Model Library. This model introduces two important basic concepts that apply to both applied physics and applied modeling: axisymmetric geometry (cylindrical) modeling and heat transfer modeling.

NOTE It is important for the new modeler to personally build each model presented within the text. There is no substitute in the path to an understanding of the modeling process for the hands-on experience of actually building, meshing, solving, and postprocessing a model. Many times the inexperienced modeler will make and subsequently correct errors, thereby adding to his or her experience and fund of modeling knowledge. Even building the simplest models will expand the modeler's fund of knowledge.

Heat transfer modeling is important in physical design and applied engineering problems. Typically, the modeler desires to understand heat generation during a process and to either add heat or remove heat to achieve or maintain a desired temperature. Figure 5.2 shows a 3D rendition of the 2D axisymmetric cylinder conduction geometry, as will be modeled here. The dashed-line ellipses in Figure 5.2 indicate the 3D rotation that would need to occur to generate the 3D solid object from the 2D cross section shown.

2D Axisymmetric Cylinder Conduction Model

NOTE This model is derived from the COMSOL cylinder conduction model. In this model, however, the selected thermally conductive solid is niobium (Nb).[12,13] Niobium has a variety of uses—as an alloying element in steels, as an alloying element in titanium turbine blades, in superconductors, as an anticorrosion coating, as an optical coating, and as an alloy in coinage.

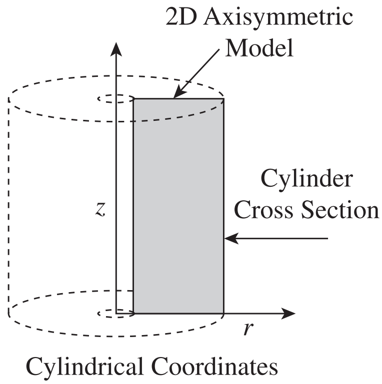


FIGURE 5.2 3D rendition of the 2D axisymmetric cylinder conduction model

To start building the Cylinder_Conduction_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Next select Heat Transfer Module > General Heat Transfer > Steady-state analysis. See Figure 5.3. Click OK.

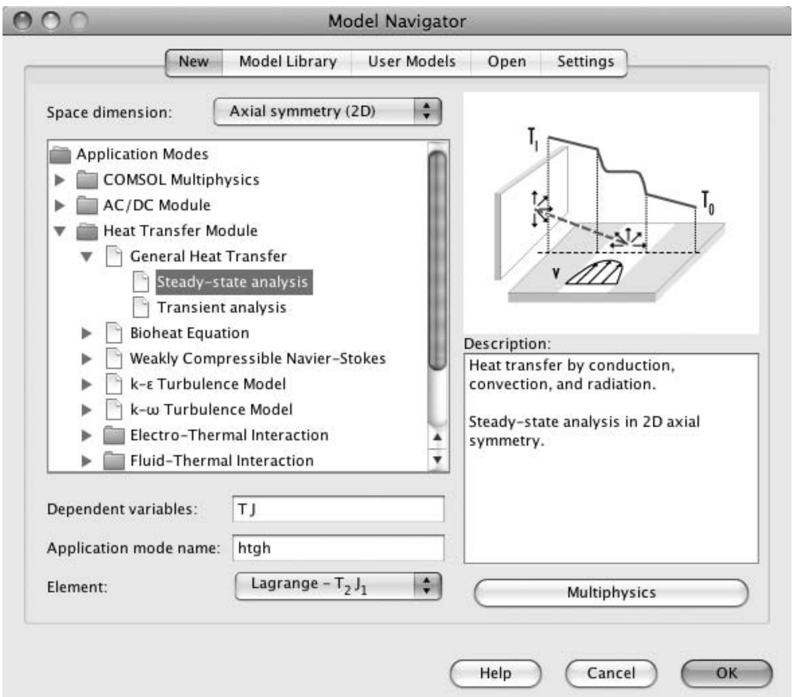


FIGURE 5.3 2D axisymmetric Cylinder_Conduction_1 Model Navigator setup

Table 5.1 Constants Edit Window

Name	Expression	Description
k_Nb	52.335[W/(m*K)]	Thermal conductivity Nb
rho_Nb	8.57e3[kg/m^3]	Density Nb
Cp_Nb	2.7e2[J/(kg*K)]	Heat capacity of Nb
T_0	2.7315e2[K]	Boundary temperature
q_0	5e5[W/m^2]	Heat flux

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.1; also see Figure 5.4. Click OK.

NOTE When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Since the settings in Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.08 and a height of 0.14. Select "Base: Corner" and set r equal to 0.02 and z equal to 0 in the Rectangle edit window. See Figure 5.5.

Click OK, and then click the Zoom Extents button. See Figure 5.6.

Using the menu bar, select Draw > Specify Objects > Point. In the Point edit window, enter the information shown in Table 5.2. See Figure 5.7.

Click OK. See Figure 5.8.

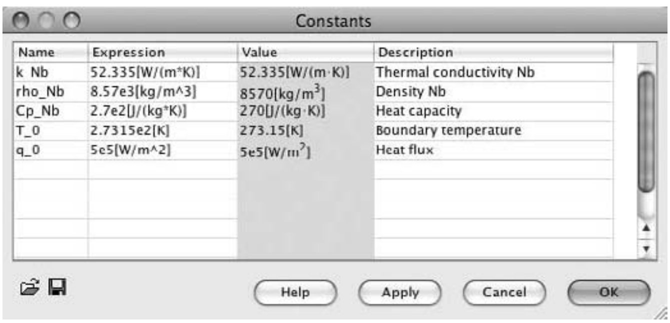


FIGURE 5.4 2D axisymmetric Cylinder_Conduction_1 model Constants edit window

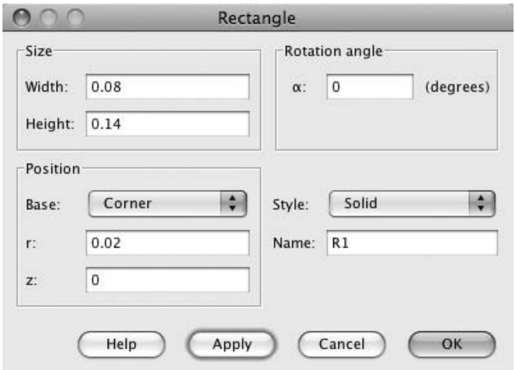


FIGURE 5.5 2D axisymmetric Cylinder_Conduction_1 model Rectangle edit window

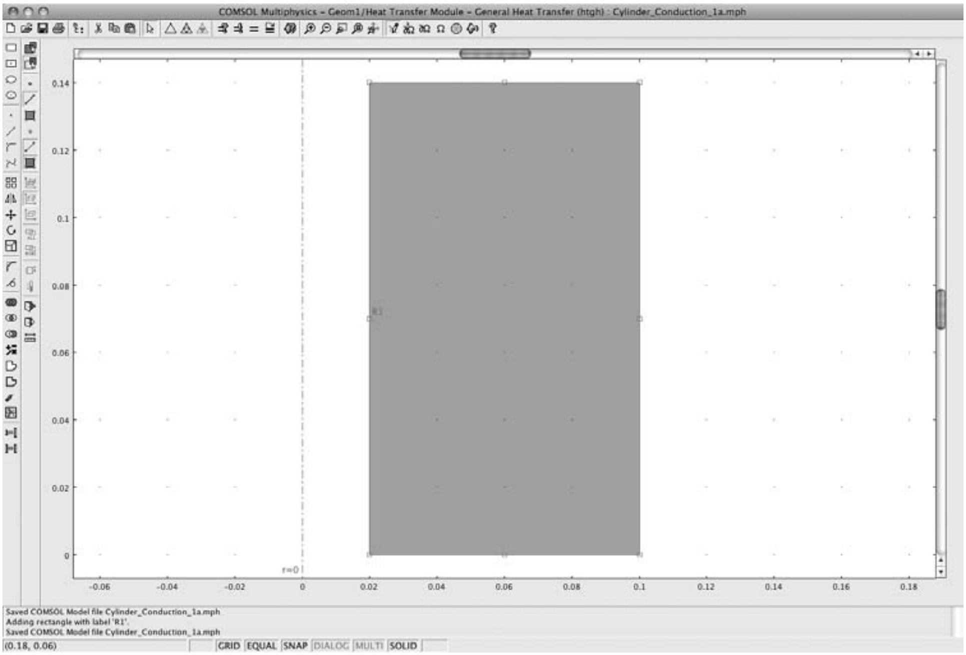


FIGURE 5.6 2D axisymmetric Cylinder_Conduction_1 model cylinder rectangle

Table 5.2 Point Edit Window

Name	*r* Location	*z* Location
Point 1	0.02	0.04
Point 2	0.02	0.1

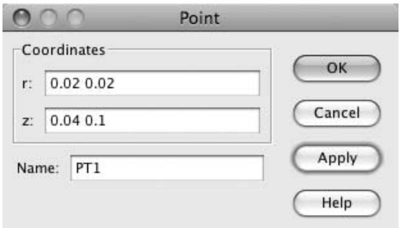


FIGURE 5.7 2D axisymmetric Cylinder_Conduction_1 model Point edit window

---

NOTE Two points have been added on the interior (small *r* value) boundary of the rectangle (cylinder cross section) to define the upper (larger *z* value) and lower (smaller *z* value) bounds of the heat-flux application region.

Physics Subdomain Settings: General Heat Transfer

Having established the geometry for the 2D axisymmetric Cylinder_Conduction_1 model (a rectangle with two points on the boundary), the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain).

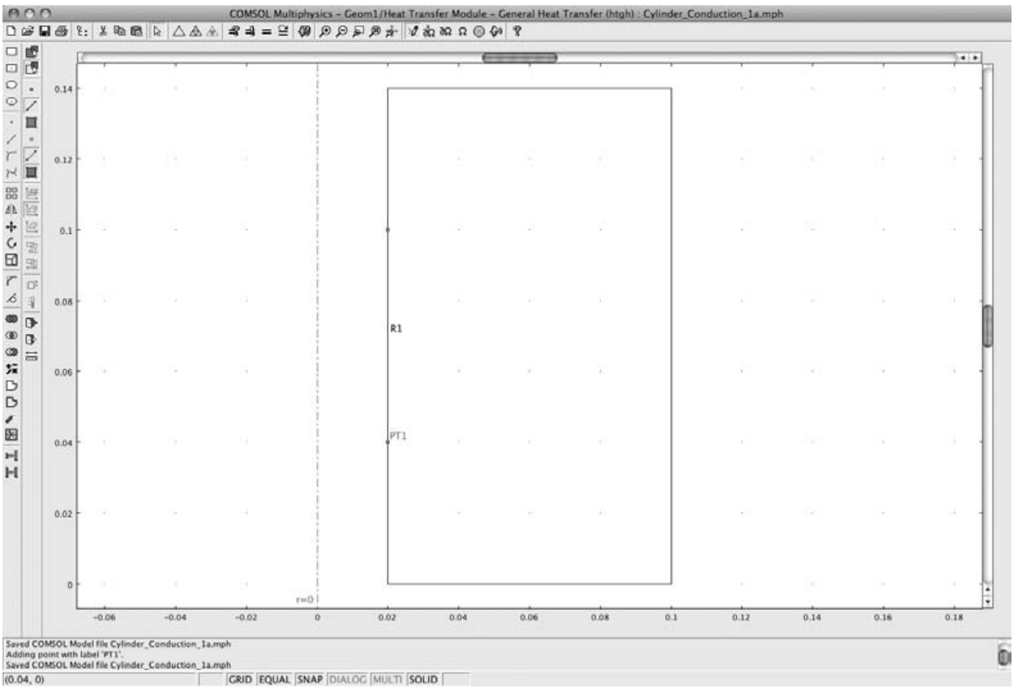


FIGURE 5.8 2D axisymmetric Cylinder_Conduction_1 model rectangle with points

Table 5.3 Subdomain Edit Windows

Name	Expression	Description
k (isotropic)	k_Nb	Thermal conductivity
ρ	rho_Nb	Density
C_P	Cp_Nb	Heat capacity

In the Subdomain edit windows, enter the information shown in Table 5.3; also see Figure 5.9. Click OK.

NOTE For static and quasi-static calculations, the only physical property value required for the conduction calculation is *k* (k_Nb). From the point of view of physical consistency, however, the density (rho_Nb) and the heat capacity (Cp_Nb) should be included as well. If *C*p and rho are set to zero, the implication is the model includes a perfect vacuum, which is logically inconsistent with the stated value of *k*. Also, by including the values for *C*p and rho in this location, they are conveniently available should the modeler wish to modify the model for transient analysis.

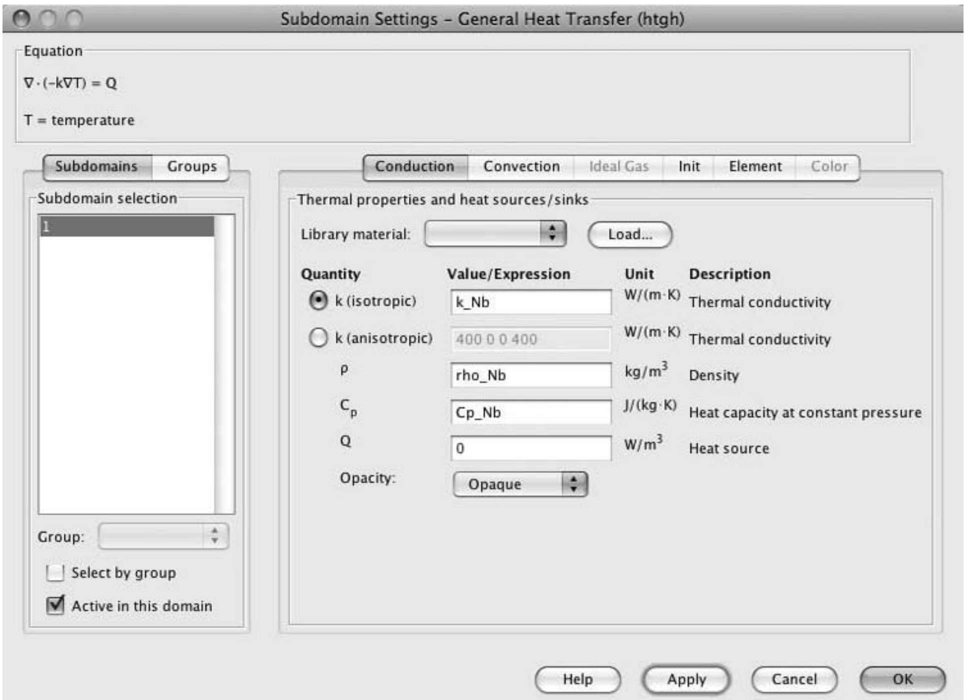


FIGURE 5.9 2D axisymmetric Cylinder_Conduction_1 model Subdomain Settings edit window

Table 5.4 Boundary Settings–General Heat Transfer Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
1, 4	Thermal insulation	—	5.10
2, 5, 6	Temperature	T_0	5.11
3	Heat flux	q_0	5.12

Physics Boundary Settings: General Heat Transfer

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value as shown in Table 5.4. Click OK. See Figures 5.10, 5.11, and 5.12.

Mesh Generation

On the toolbar, click the Initialize Mesh button once. Click the Refine Mesh button once. This results in a mesh of approximately 2200 elements. See Figure 5.13.

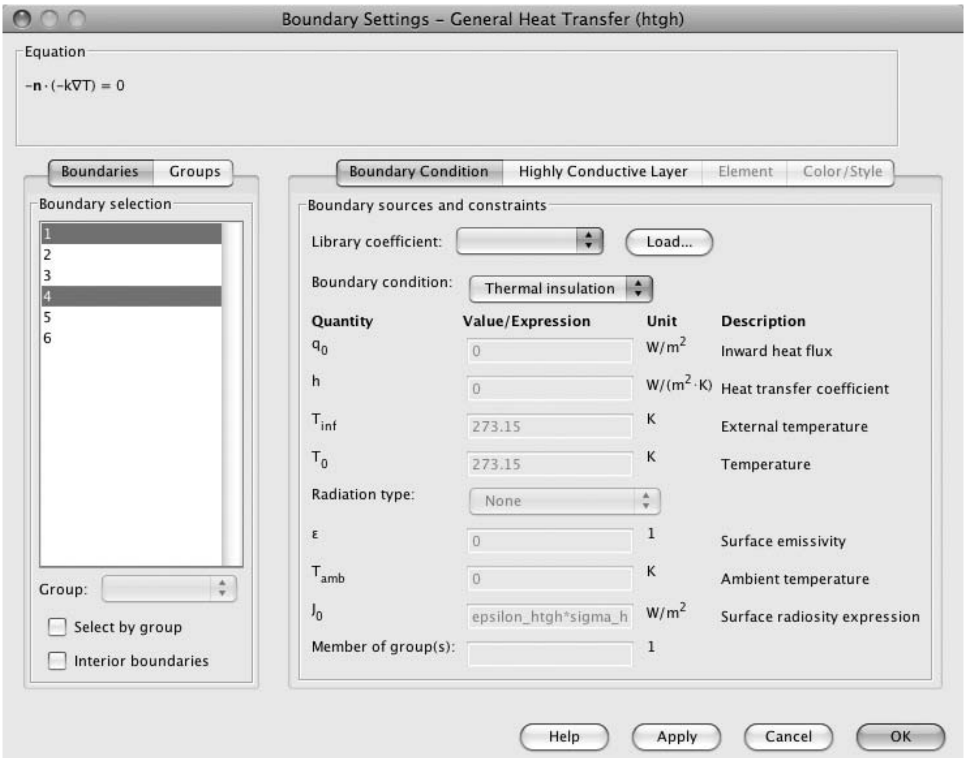


FIGURE 5.10 2D axisymmetric Cylinder_Conduction_1 model Boundary Settings (1, 4) edit window

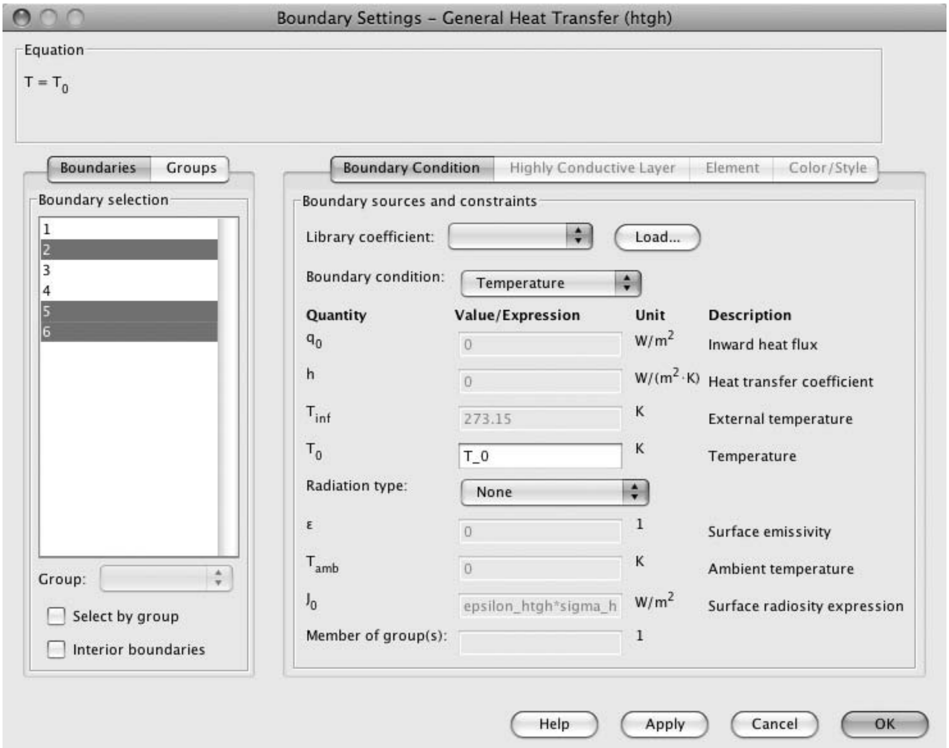


FIGURE 5.11 2D axisymmetric Cylinder_Conduction_1 model Boundary Settings (2, 5, 6) edit window

Solving the 2D Axisymmetric Cylinder_Conduction_1 Model

Using the menu bar, select Solve > Solve Problem. The COMSOL Multiphysics software automatically selects the Stationary Solver.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, always change the chosen solver or the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Figure 5.14 shows the modeling solution results obtained using single-value parameters with the default solver (UMFPACK).

Parametric Solving of the 2D Axisymmetric Cylinder_Conduction_1 Model

Now that the model has been built, it is relatively easy to expand the model to calculate other quasi-static solutions.

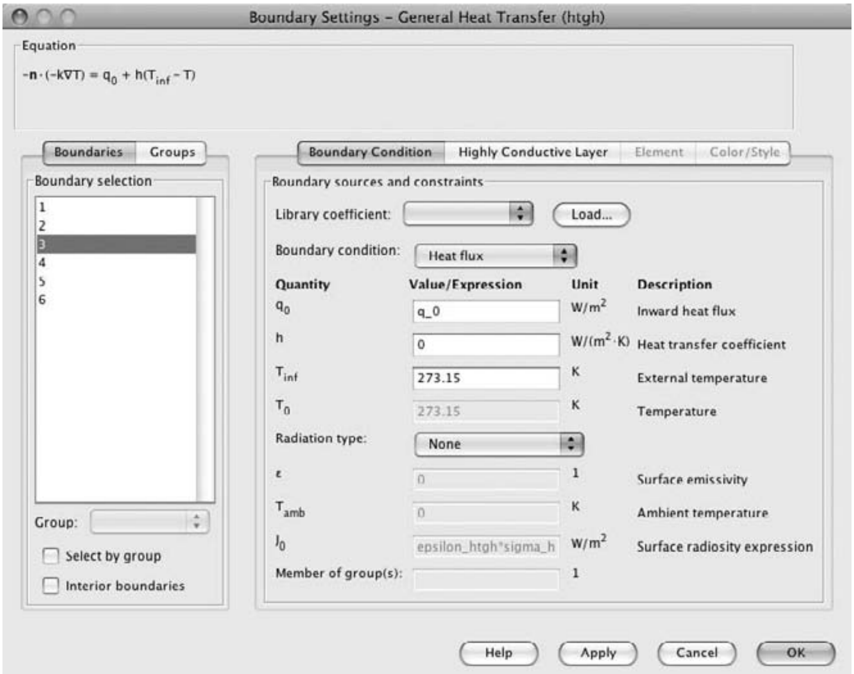


FIGURE 5.12 2D axisymmetric Cylinder_Conduction_1 model Boundary Settings (3) edit window

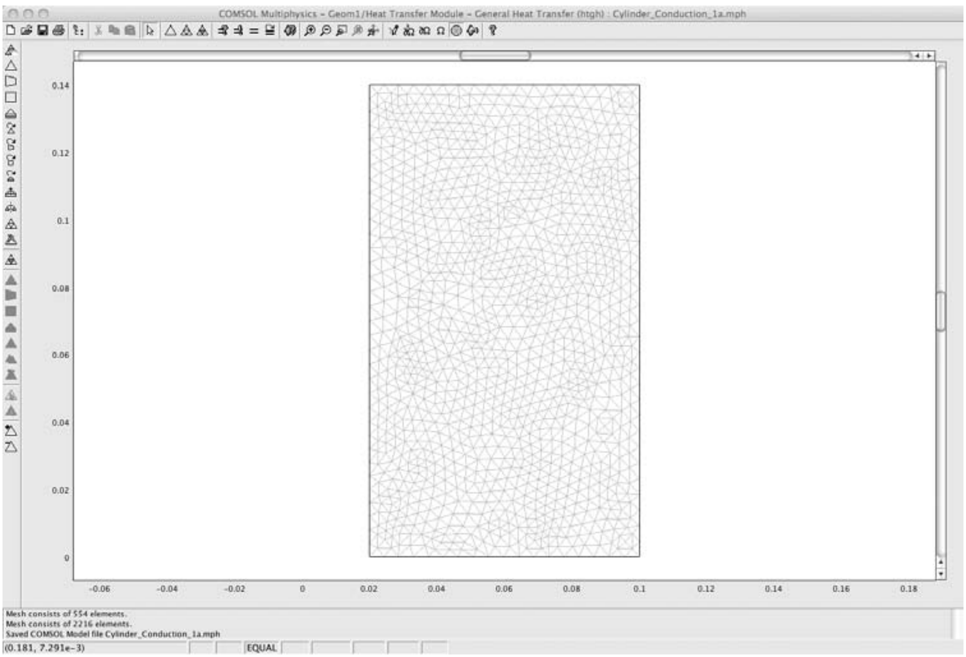


FIGURE 5.13 2D axisymmetric Cylinder_Conduction_1 model Mesh window

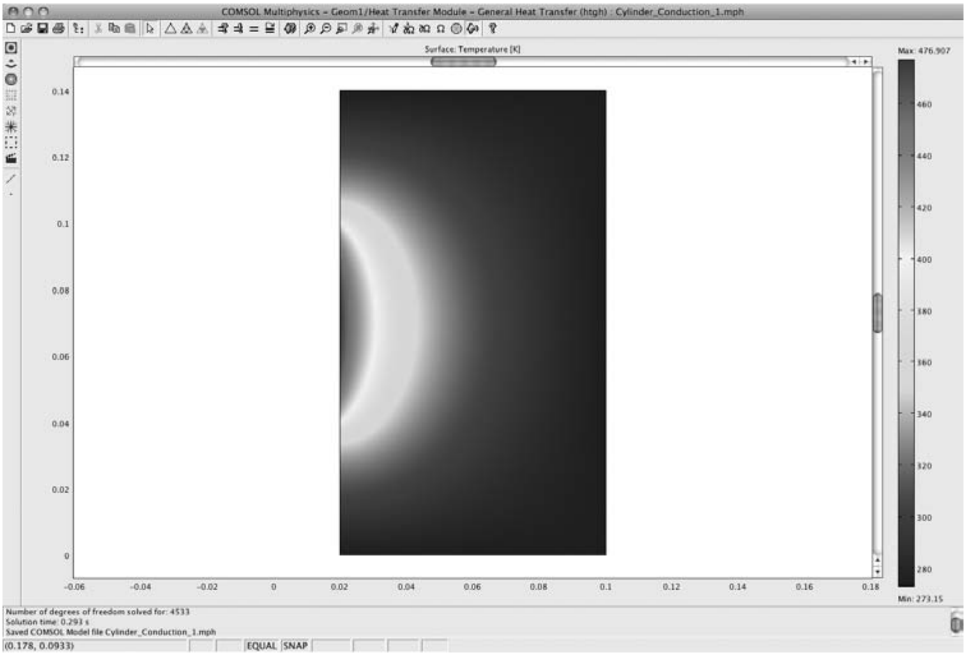


FIGURE 5.14 2D axisymmetric Cylinder_Conduction_1 model using the default solver

---

NOTE This time, instead of using the default solver, the model is run using multiple value parameters in the Parametric Solver (UMFPACK) as the initial solver. The Parametric Solver (UMFPACK) results include not only the default solution, but also solutions at a number of other values of heat flux (q_0).

Using the menu bar, select File > Save as. Enter Cylinder_Conduction_1p.

From the menu bar, select File > Reset Model >Yes. On the menu bar, click the Initialize Mesh button once. Click the Refine Mesh button once. This results in a mesh of approximately 2200 elements.

From the menu bar, select Solve > Solver Parameters > Parametric. Enter q_0 in the Parameter name edit window. Enter 0:5e3:5e5 in the Parameter values edit window. See Figure 5.15. Click OK.

From the menu bar, select Solve > Solve Problem. See Figure 5.16.

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.17.

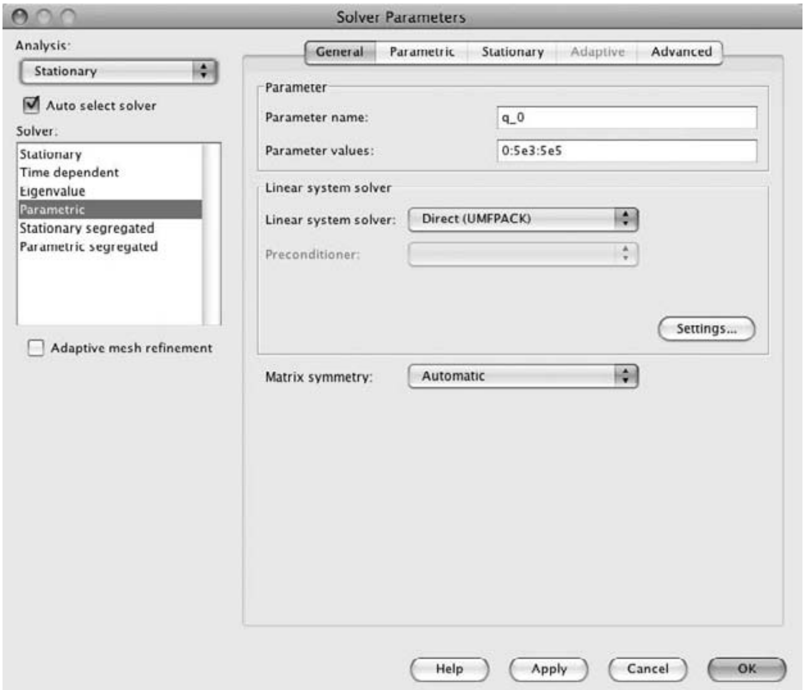


FIGURE 5.15 2D axisymmetric Cylinder_Conduction_1p model Solver Parameters edit window

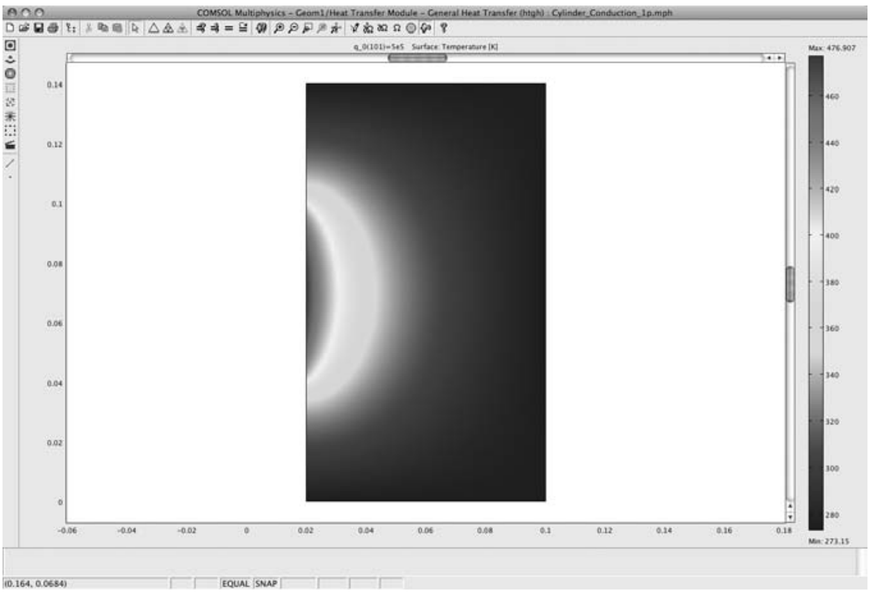


FIGURE 5.16 2D axisymmetric Cylinder_Conduction_1p model using the Parametric Solver (UMFPACK)

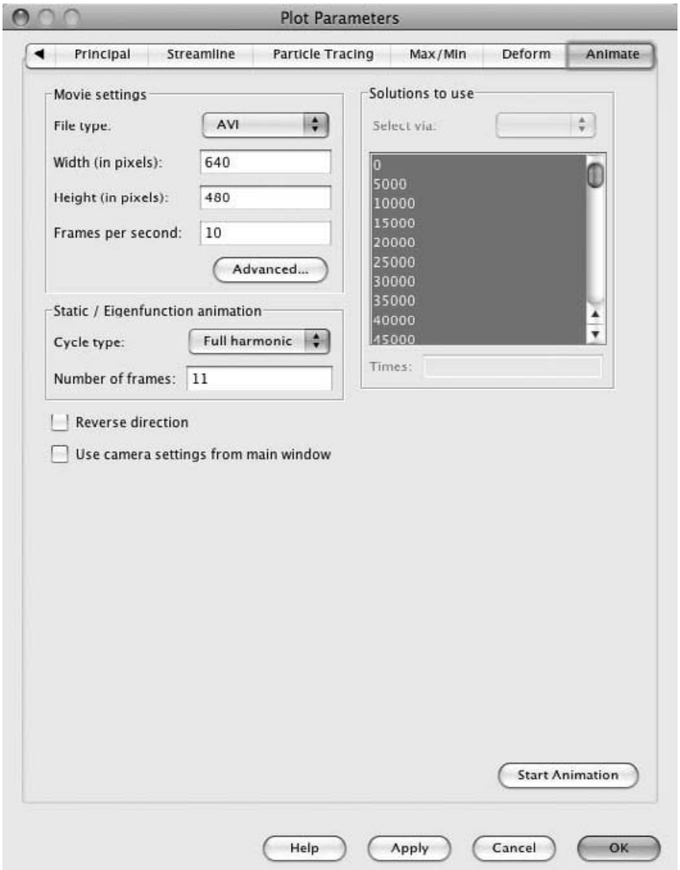


FIGURE 5.17 2D axisymmetric Cylinder_Conduction_1p model Plot Parameters window

Click the Start Animation button. See Figure 5.18.

Alternatively, you can play the file Movie5_CC_1p.avi that was supplied with this book.

First Variation on the 2D Axisymmetric Cylinder Conduction Model

NOTE This model is derived from the COMSOL cylinder conduction model. This model, however, is built using the basic COMSOL Multiphysics software package, instead of the Heat Transfer Module. The selected thermally conductive solid is, as in the initial model, niobium (Nb). The modeler should note, as mentioned previously, that for static and quasi-static calculations, the only physical property value required for the thermal conduction calculation is *k* (k_Nb), the thermal conductivity. That property is the only one that will be used in this model.

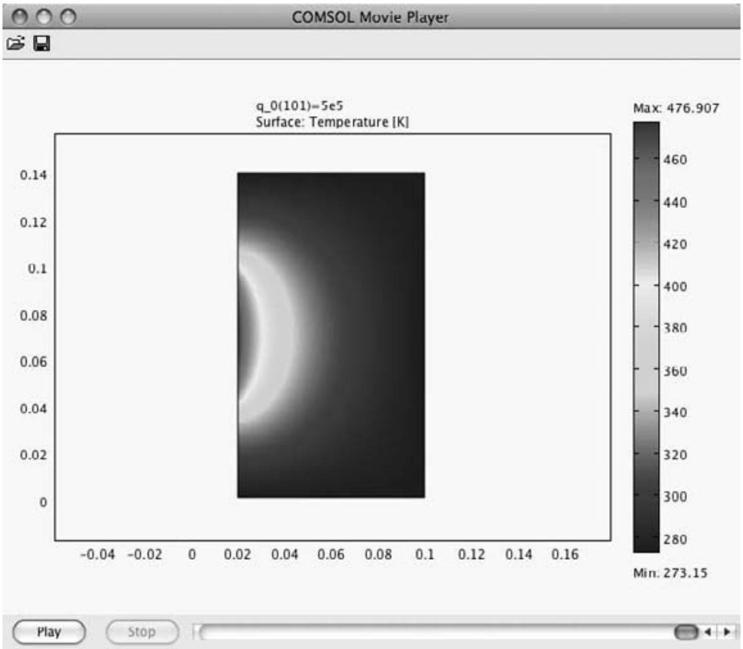


FIGURE 5.18 2D axisymmetric Cylinder_Conduction_1p model animation, final frame

To start building the Cylinder_Conduction_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select COMSOL Multiphysics > Heat Transfer > Conduction > Steady-state analysis. See Figure 5.19. Click OK.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.5; also see Figure 5.20. Click OK.

NOTE When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.08 and a height of 0.14. Select "Base: Corner" and set r equal to 0.02 and z equal to 0 in the Rectangle edit window. See Figure 5.21.

Table 5.5 Constants Edit Window

Name	Expression	Description
k_Nb	52.335[W/(m*K)]	Thermal conductivity Nb
T_0	2.7315e2[K]	Boundary temperature
q_0	5e5[W/m^2]	Heat flux

FIGURE 5.19 2D axisymmetric Cylinder_Conduction_2 Model Navigator setup

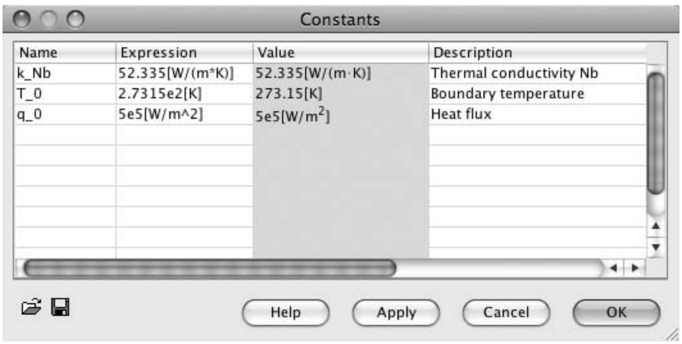


FIGURE 5.20 2D axisymmetric Cylinder_Conduction_2 model Constants edit window

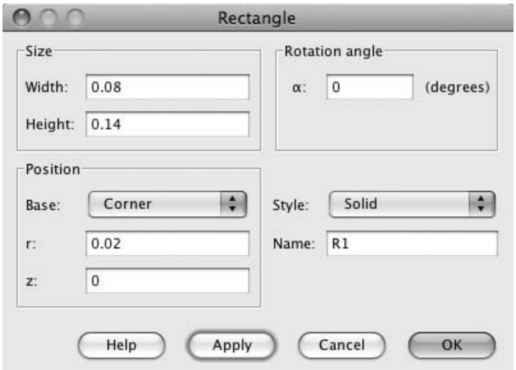


FIGURE 5.21 2D axisymmetric Cylinder_Conduction_2 model Rectangle edit window

Click OK, and then click the Zoom Extents button. See Figure 5.22.

Using the menu bar, select Draw > Specify Objects > Point. In the Point edit window, enter the information shown in Table 5.6. See Figure 5.23.

Click OK. See Figure 5.24.

NOTE Two points have been added, as in the earlier model, on the interior (small *r* value) boundary of the rectangle (cylinder cross section) to define the upper (larger *z* value) and lower (smaller *z* value) bounds of the heat-flux application region.

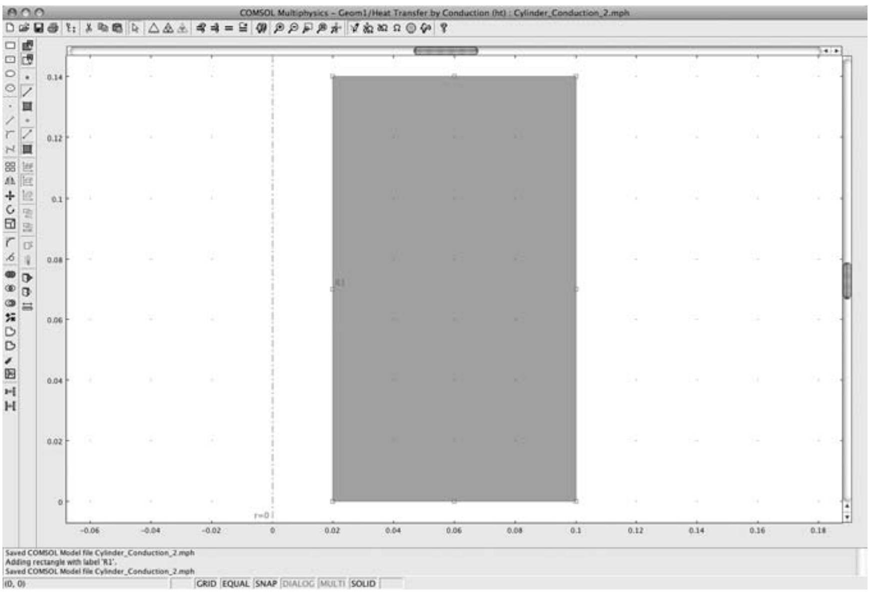


FIGURE 5.22 2D axisymmetric Cylinder_Conduction_2 model cylinder rectangle

Table 5.6 Point Edit Window

Name	*r* Location	*z* Location
Point 1	0.02	0.04
Point 2	0.02	0.1

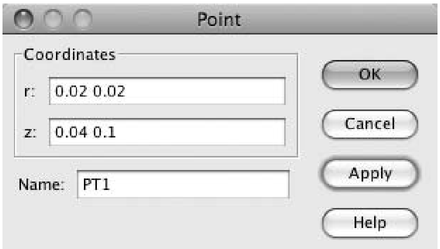


FIGURE 5.23 2D axisymmetric Cylinder_Conduction_2 model Point edit window

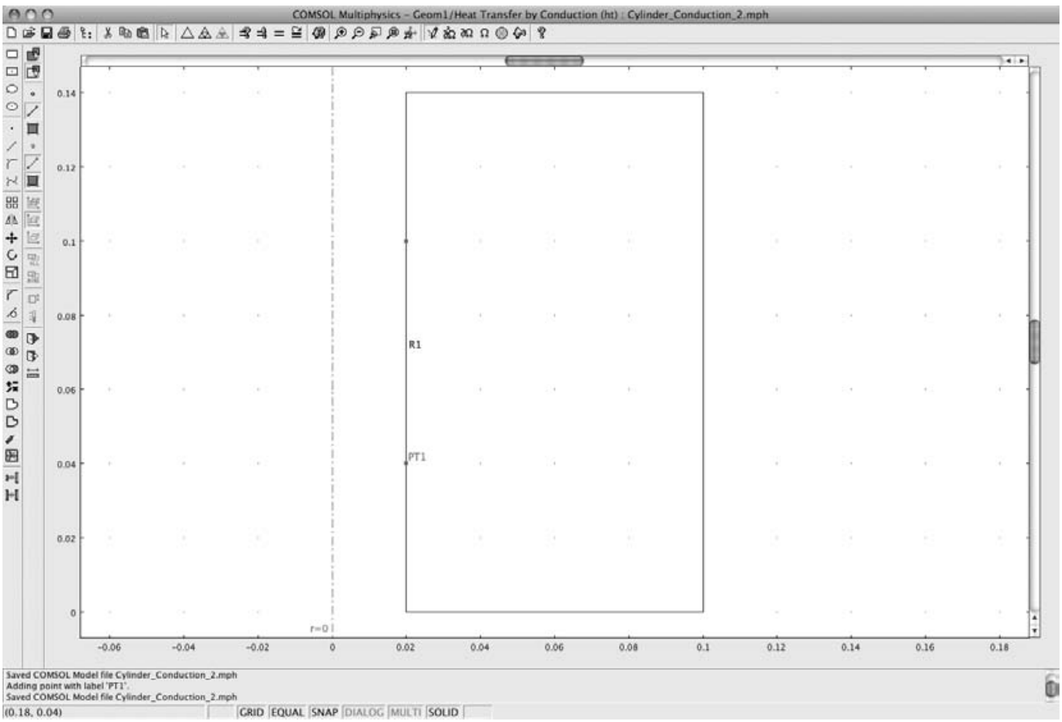


FIGURE 5.24 2D axisymmetric Cylinder_Conduction_2 model rectangle with points

Table 5.7 Subdomain Edit Window

Name	Expression	Description
k (isotropic)	k_Nb	Thermal conductivity

Physics Subdomain Settings: General Heat Transfer

Having established the geometry for the 2D axisymmetric Cylinder_Conduction_2 model (a rectangle with two points on the boundary), the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). In the Subdomain edit windows, enter the information shown in Table 5.7; also see Figure 5.25. Click OK.

NOTE For static and quasi-static calculations, the only physical property value required for the calculation is *k* (k_Nb).

Physics Boundary Settings: General Heat Transfer

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 5.8. Click OK. See Figures 5.26, 5.27, and 5.28.

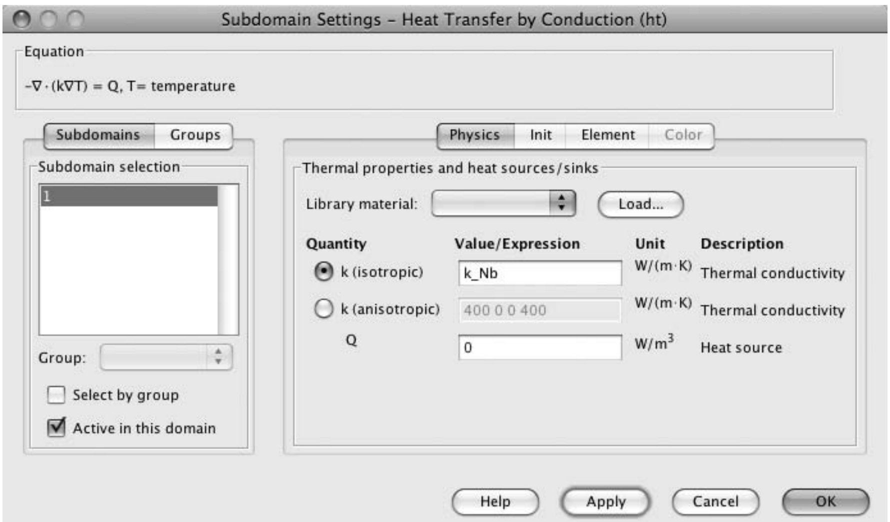


FIGURE 5.25 2D axisymmetric Cylinder_Conduction_2 Model Subdomain Settings edit window

Table 5.8 Boundary Settings–General Heat Transfer Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
1, 4	Thermal insulation	—	5.26
2, 5, 6	Temperature	T_0	5.27
3	Heat flux	q_0	5.28

Mesh Generation

On the toolbar, click the Initialize Mesh button once. Click the Refine Mesh button once. This results in a mesh of approximately 2200 elements. See Figure 5.29.

Solving the 2D Axisymmetric Cylinder_Conduction_2 Model

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, always change the chosen solver and the parametric settings. This time, instead of using the default solver, the model is run using the Parametric Solver (UMFPACK) as the initial solver. The Parametric Solver (UMFPACK) results include not only the default solution, but also solutions at a number of other values of heat flux (q_0).

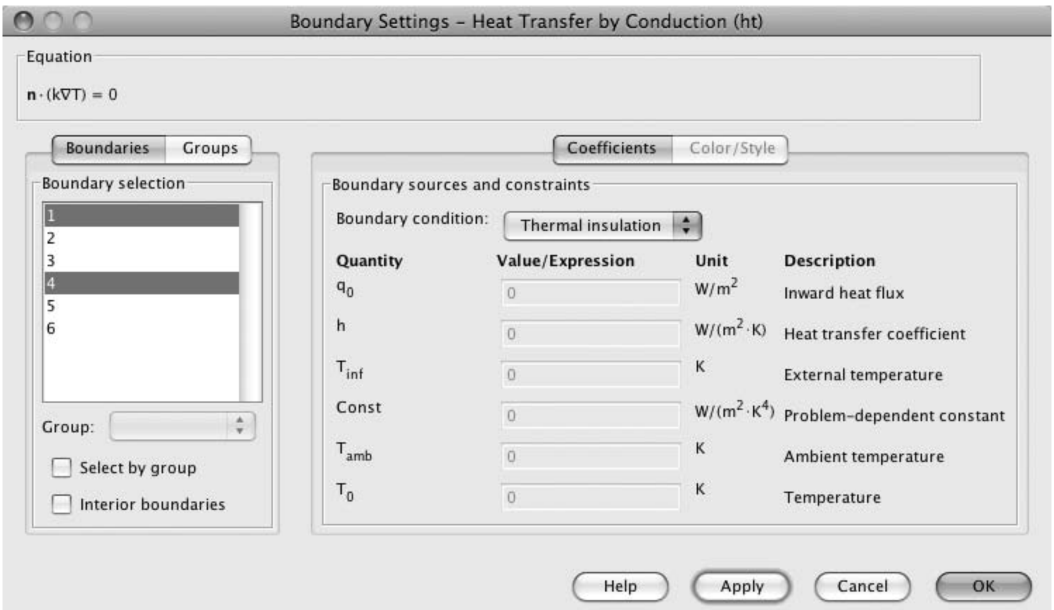


FIGURE 5.26 2D axisymmetric Cylinder_Conduction_2 model Boundary Settings (1, 4) edit window

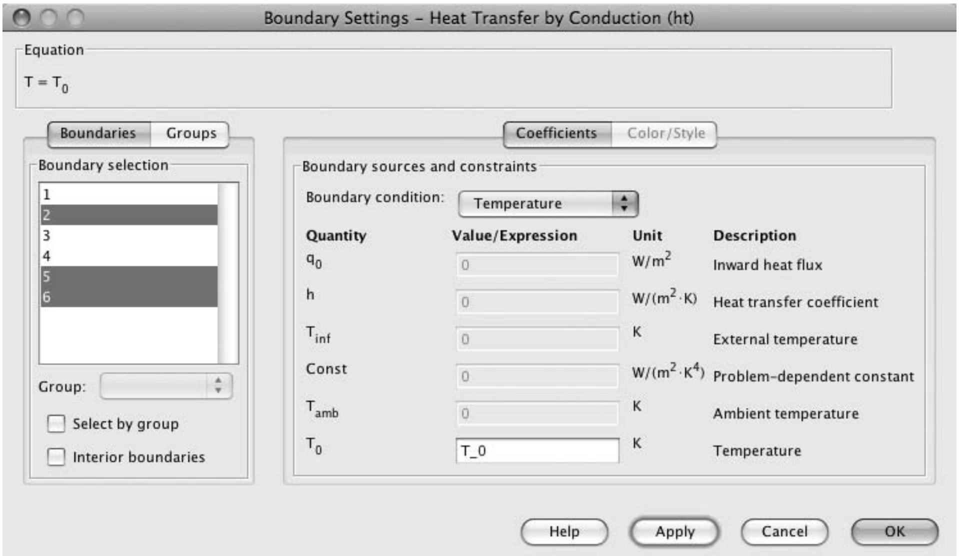


FIGURE 5.27 2D axisymmetric Cylinder_Conduction_2 model Boundary Settings (2, 5, 6) edit window

Using the menu bar, select Solve > Solver Parameters > Parametric. Enter q_0 in the Parameter name edit window. Enter 0:5e3:5e5 in the Parameter values edit window. See Figure 5.30. Click OK.

From the menu bar, select Solve > Solve Problem. See Figure 5.31.

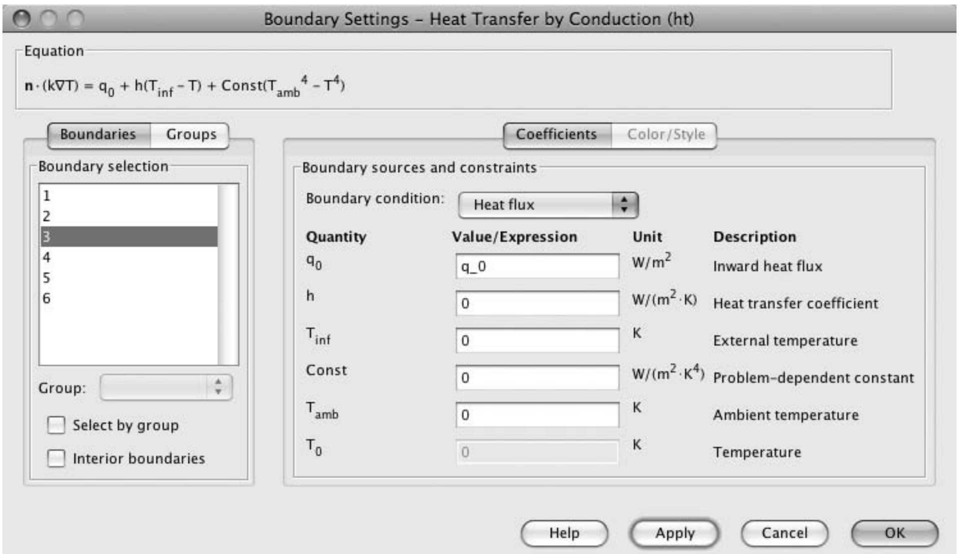


FIGURE 5.28 2D axisymmetric Cylinder_Conduction_2 model Boundary Settings (3) edit window

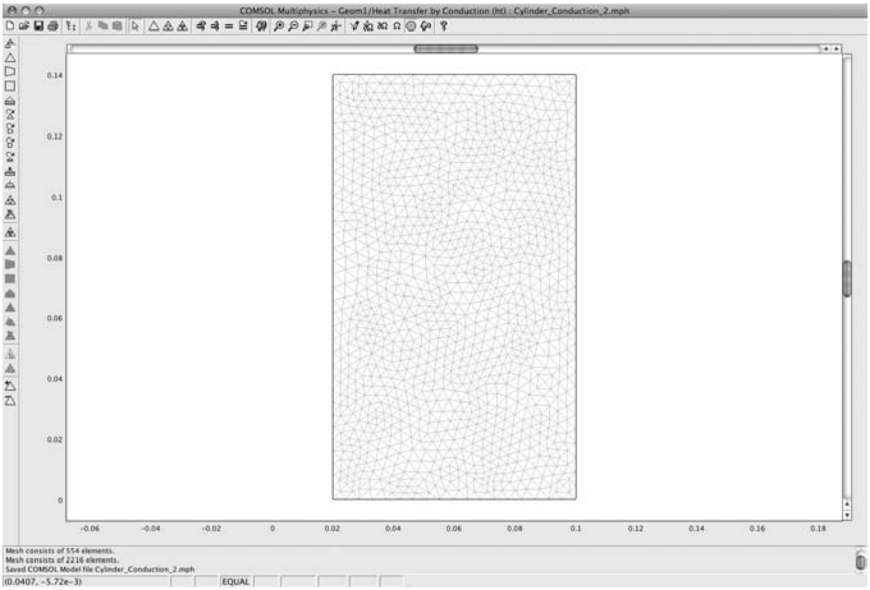


FIGURE 5.29 2D axisymmetric Cylinder_Conduction_2 model mesh window

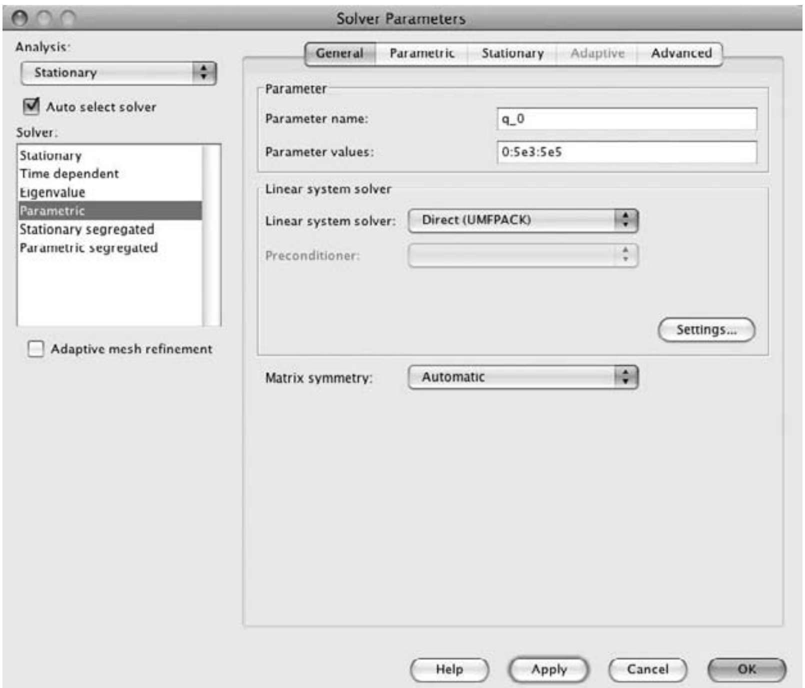


FIGURE 5.30 2D axisymmetric Cylinder_Conduction_2 model Solver Parameters edit window

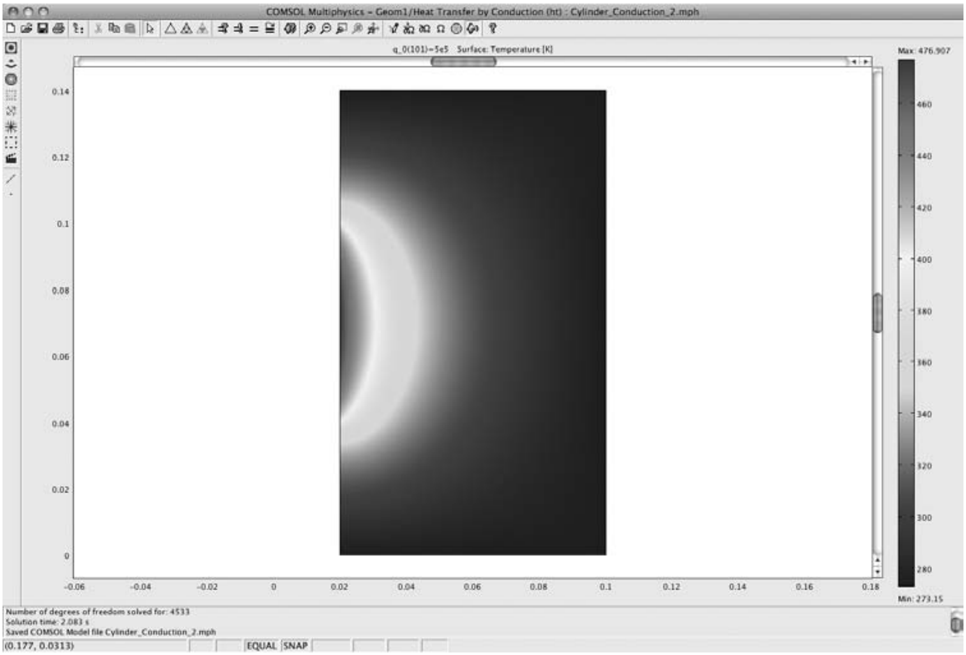


FIGURE 5.31 2D axisymmetric Cylinder_Conduction_2 model using the Parametric Solver (UMFPACK)

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.32.

Click the Start Animation button. See Figure 5.33.

Alternatively, you can play the file Movie5_CC_2.avi that was supplied with this book.

Comparison of Cylinder Conduction Models 1p and 2

As can be readily seen in Table 5.9, the calculated values for Cylinder Conduction Models 1p and 2 are exactly the same for the simple conduction calculation, as would be expected. The advantage of using the Heat Transfer Module, as needed, is that it can accommodate more complex physics. See Figures 5.34 and 5.35.

Table 5.9 Comparison of T-max for Cylinder Conduction Models 1p and 2

Model Number	Module Used	T-max	Figure Number
1p	Heat Transfer Module	476.907 K	5.34
2	Basic Heat Transfer	476.907 K	5.35

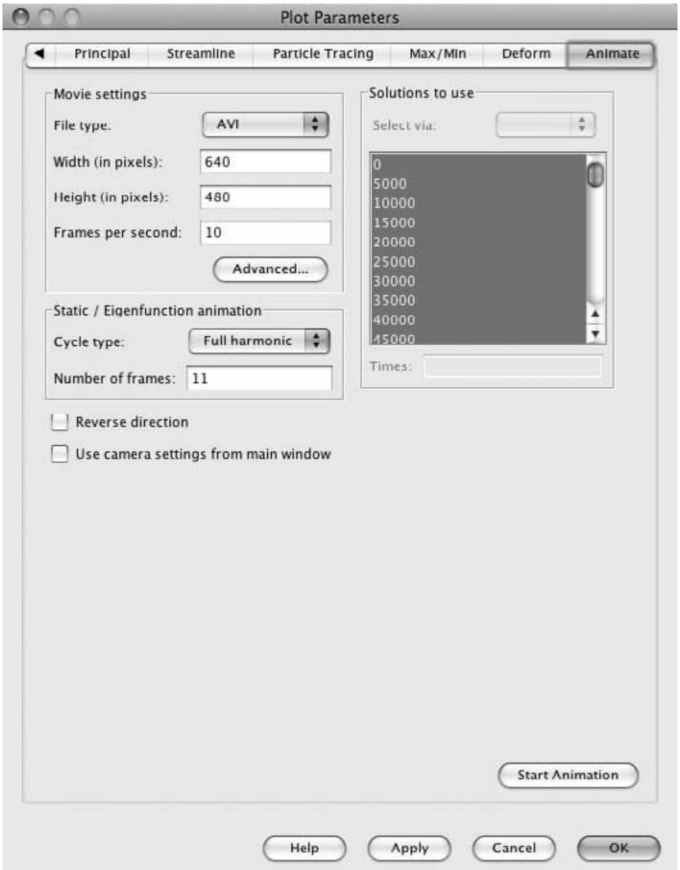


FIGURE 5.32 2D axisymmetric Cylinder_Conduction_2 model Plot Parameters window

Second Variation on the 2D Axisymmetric Cylinder Conduction Model, Including a Vacuum Cavity

NOTE This model is derived from the COMSOL cylinder conduction model. In this model, the selected thermally conductive solid is niobium (Nb).[12,13] A vacuum cavity has been added to the cylinder geometry. With the added vacuum cavity, the modeler can explore some of the additional heat transfer modeling capabilities of the Heat Transfer Module. Vacuum isolation is a valuable tool in lowering heat loss in modern machines.

To start building the Cylinder_Conduction_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > General Heat Transfer > Steady-state analysis. See Figure 5.36. Click OK.

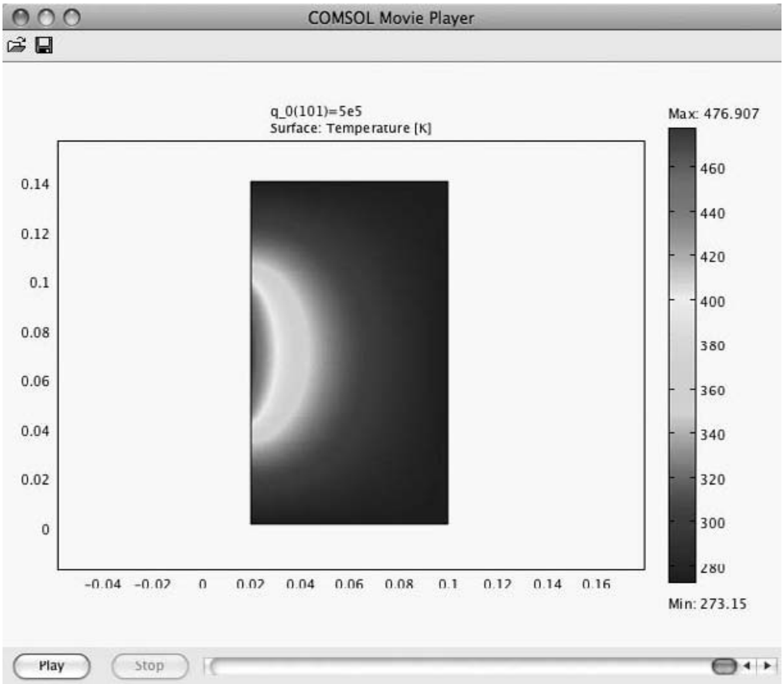


FIGURE 5.33 2D axisymmetric Cylinder_Conduction_2 model animation, final frame

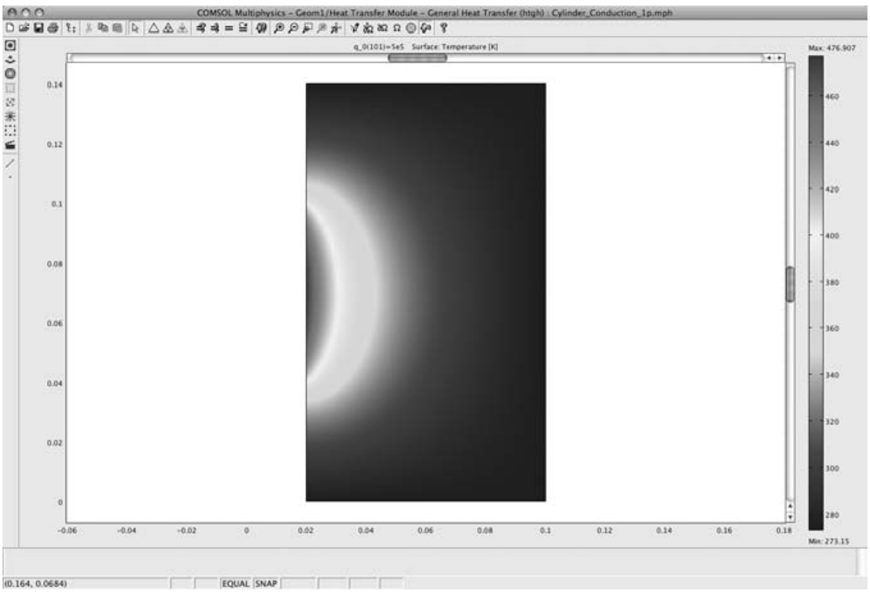


FIGURE 5.34 2D axisymmetric Cylinder_Conduction_1p model, final frame

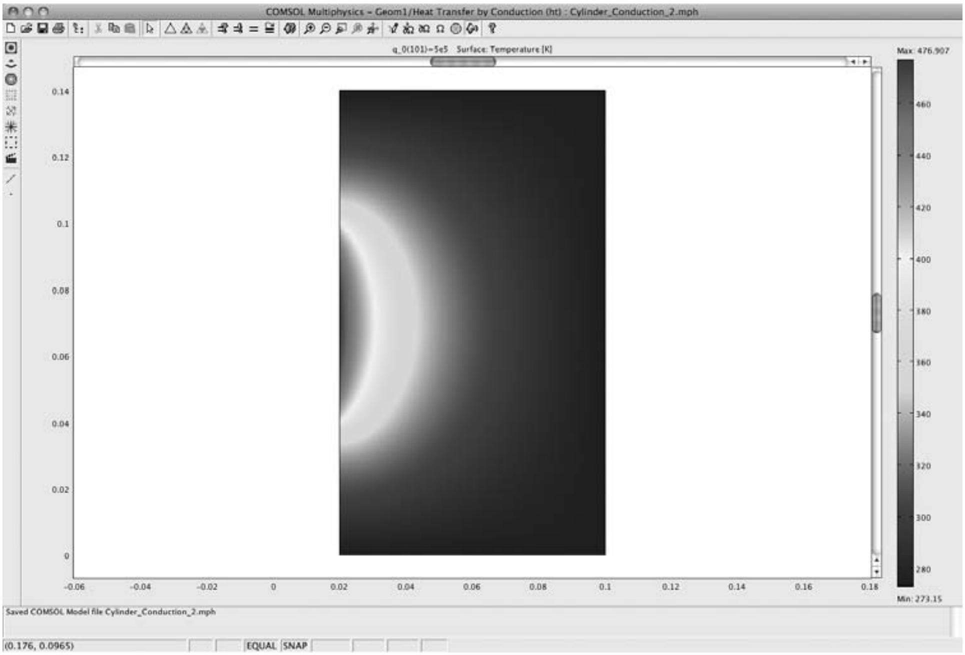


FIGURE 5.35 2D axisymmetric Cylinder_Conduction_2 model, final frame

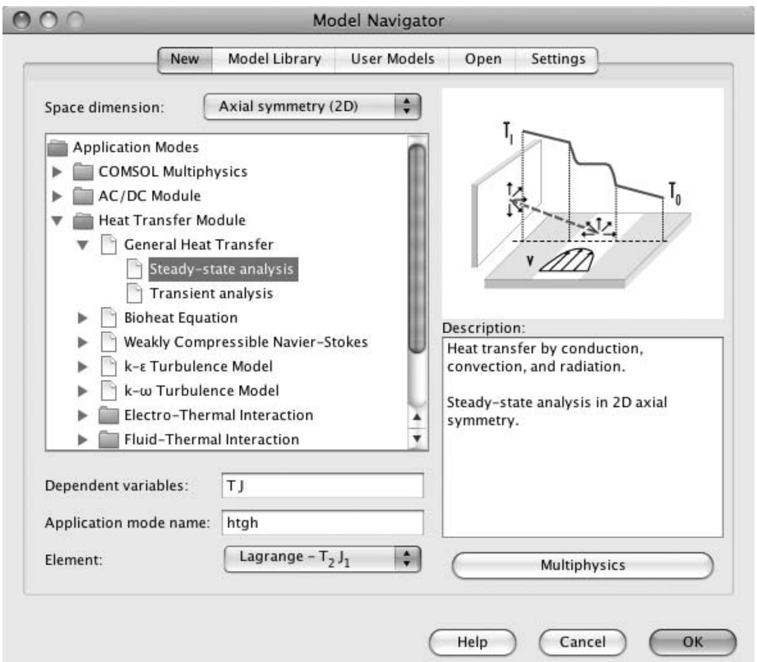


FIGURE 5.36 2D axisymmetric Cylinder_Conduction_3 Model Navigator setup

Table 5.10 Constants Edit Window

Name	Expression	Description
k_Nb	52.335[W/(m*K)]	Thermal conductivity Nb
rho_Nb	8.57e3[kg/m^3]	Density Nb
Cp_Nb	2.7e2[J/(kg*K)]	Heat capacity Nb
T_0	2.7315e2[K]	Boundary temperature
q_0	5e5[W/m^2]	Heat flux
p_0	1.33e-7[Pa]	Pressure in vacuum

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.10; also see Figure 5.37. Click OK.

NOTE When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Draw > Specify Objects > Rectangle. Create the two rectangles indicated in Table 5.11.

Table 5.11 Rectangle Edit Window

Rectangle Number	Width	Height	Base	*r*	*z*	Figure Number
1	0.08	0.14	Corner	0.02	0	5.38
2	0.002	0.139	Corner	0.06	0.0005	5.39

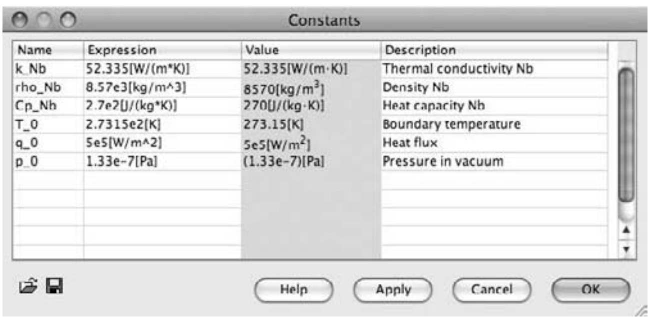


FIGURE 5.37 2D axisymmetric Cylinder_Conduction_3 model Constants edit window

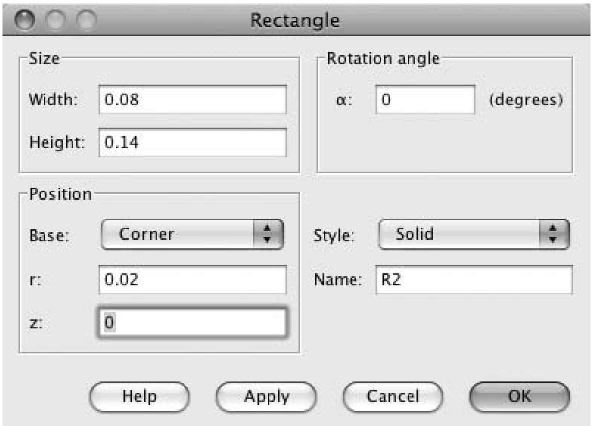


FIGURE 5.38 2D axisymmetric Cylinder_Conduction_3 model Rectangle edit window (1)

Click OK, and then click the Zoom Extents button. See Figure 5.40.

Using the menu bar, select Draw > Specify Objects > Point. In the Point edit window, enter the information shown in Table 5.12. See Figure 5.41.

Click OK. See Figure 5.42

NOTE Two points have been added on the interior (small *r* value) boundary of the rectangle (cylinder cross section) to define the upper (larger *z* value) and lower (smaller *z* value) bounds of the heat-flux application region.

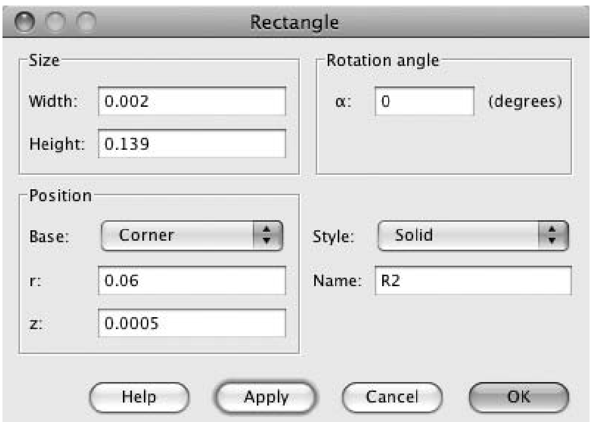


FIGURE 5.39 2D axisymmetric Cylinder_Conduction_3 model Rectangle edit window (2)

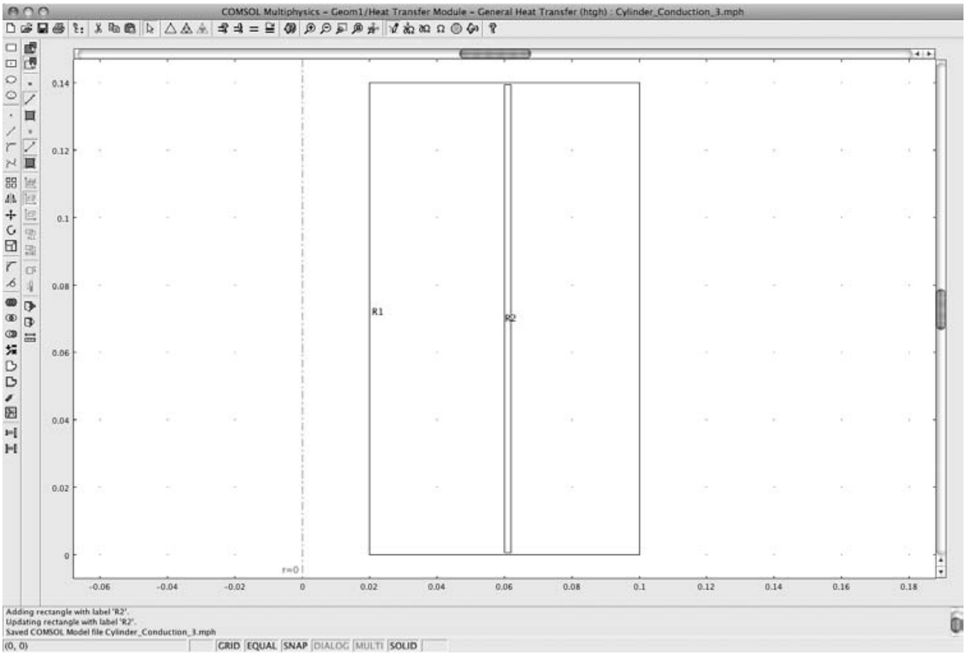


FIGURE 5.40 2D axisymmetric Cylinder_Conduction_3 model cylinder rectangles (1 and 2)

Using the menu bar, select Draw > Create Composite Object. Enter R1 + R2 in the Set formula edit window. Verify or check the Keep interior boundaries check box. See Figure 5.43.

Click OK. See Figure 5.44

Table 5.12 Point Edit Window

Name	*r* Location	*z* Location
Point 1	0.02	0.04
Point 2	0.02	0.1

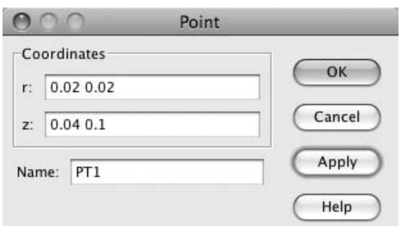


FIGURE 5.41 2D axisymmetric Cylinder_Conduction_3 model Point edit window

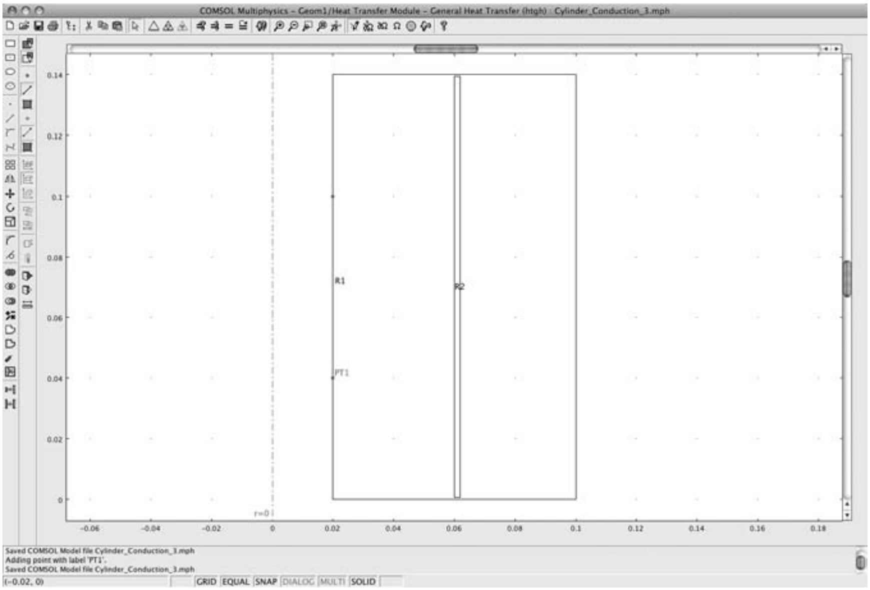


FIGURE 5.42 2D axisymmetric Cylinder_Conduction_3 model rectangles with points

Physics Subdomain Settings: General Heat Transfer

Having established the geometry for the 2D axisymmetric Cylinder_Conduction_3 model (a rectangle with two points on the boundary), the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select "subdomain 1" in the Subdomain selection window. In the Subdomain edit windows, enter the information shown in Table 5.13. See Figure 5.45.

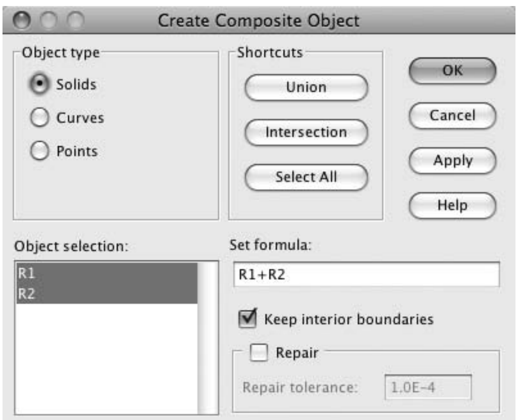


FIGURE 5.43 2D axisymmetric Cylinder_Conduction_3 model Create Composite Object edit window

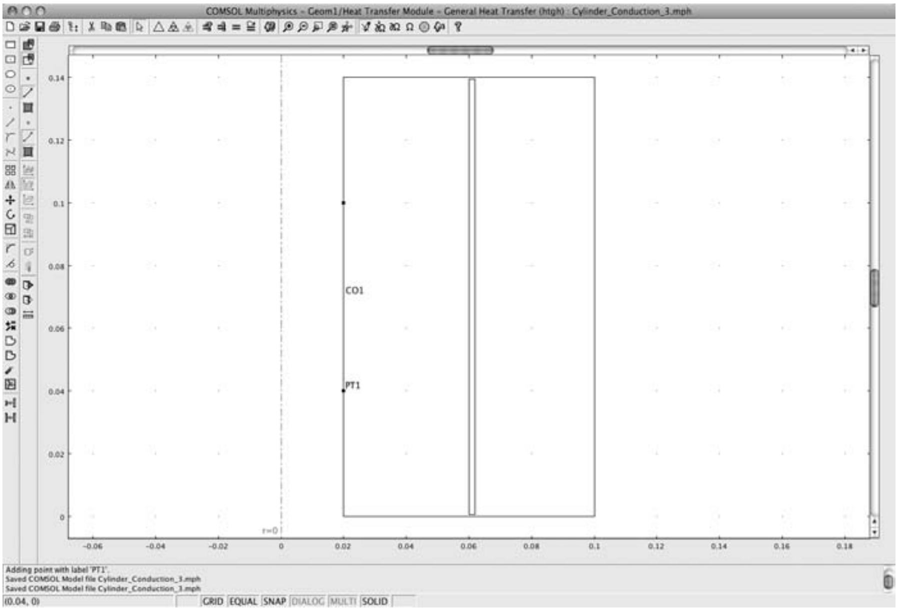


FIGURE 5.44 2D axisymmetric Cylinder_Conduction_3 model composite object

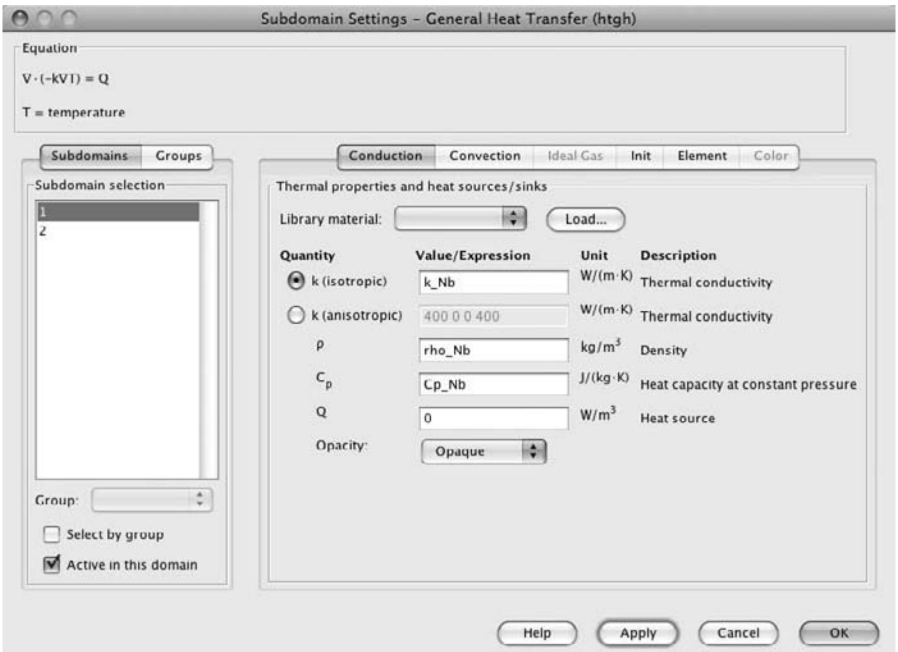


FIGURE 5.45 2D axisymmetric Cylinder_Conduction_3 model Subdomain Settings (1) edit window

Table 5.13 Subdomain Edit Window

Name	Expression	Description
k (isotropic)	k_Nb	Thermal conductivity
ρ	rho_Nb	Density
C_P	Cp_Nb	Heat capacity

Select “subdomain 2” in the Subdomain selection window. Click the Library material Load button. Select Liquids and Gases > Gases > Air, I atm. Click OK.

Enter the term p_0 in place of p in the expression rho(p…in the Density edit window. Click on the Opacity pull-down list. Select “Transparent.” See Figure 5.46. Click OK.

NOTE The insertion of p_0 into the density function for air sets the pressure in the vacuum cavity. The selection of “Transparent” allows energy transfer by radiation.

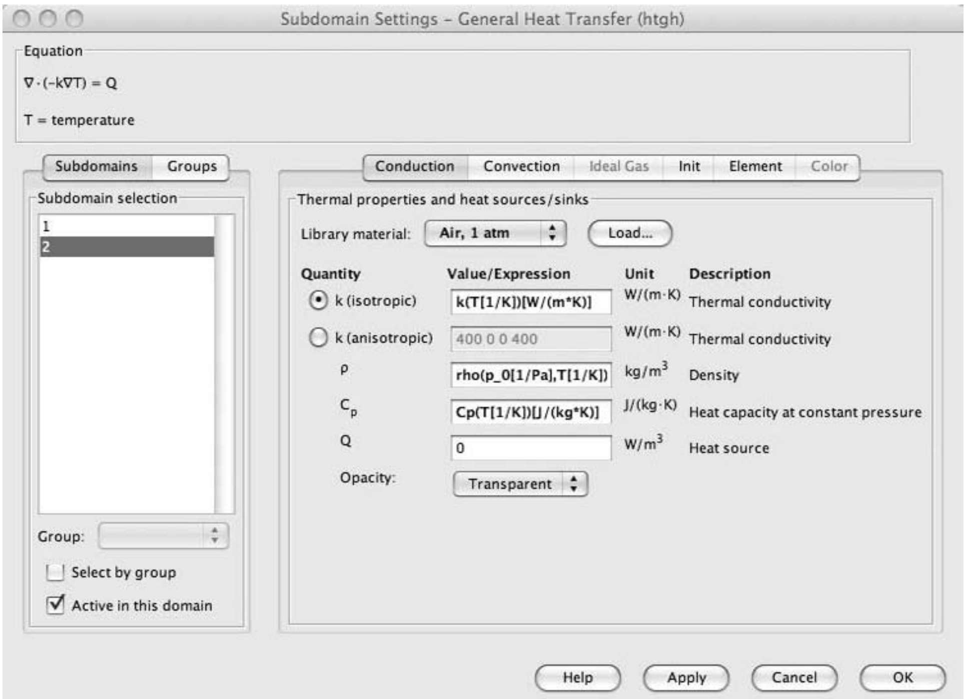


FIGURE 5.46 2D axisymmetric Cylinder_Conduction_3 model Subdomain Settings (2) edit window

Table 5.14 Boundary Settings–General Heat Transfer Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
1, 4	Thermal insulation	—	5.47
2, 5,10	Temperature	T_0	5.48
3	Heat flux	q_0	5.49

NOTE For static and quasi-static calculations, the only physical property value required for the conduction calculation is *k* (k_Nb). However, from the point of view of physical consistency, the density (rho_Nb) and the heat capacity (Cp_Nb) should be included. If *C*p and rho are set to zero, the implication is that of a perfect vacuum, which is logically inconsistent with the stated value of *k*. Also, by including the values for *C*p and rho in this location, they are conveniently available should the modeler wish to modify the model for transient analysis.

Physics Boundary Settings: General Heat Transfer

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 5.14. Click OK. See Figures 5.47, 5.48, and 5.49.

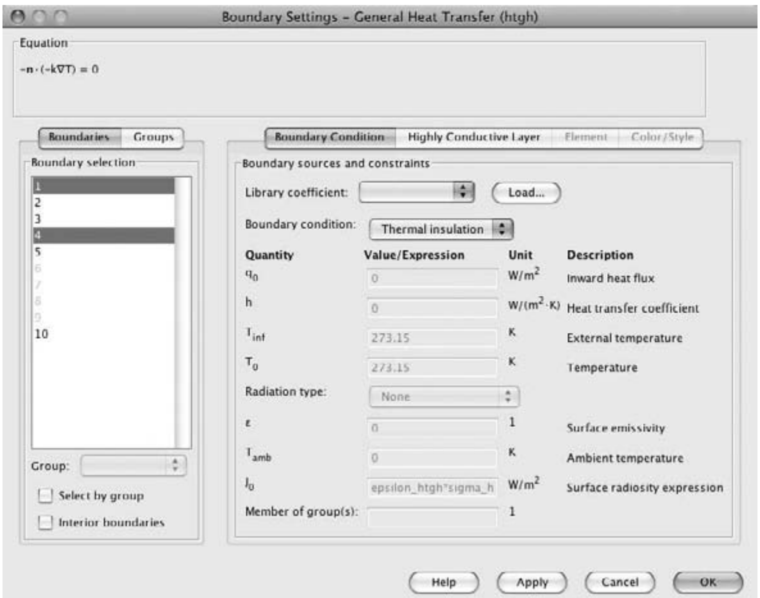


FIGURE 5.47 2D axisymmetric Cylinder_Conduction_3 model Boundary Settings (1, 4) edit window

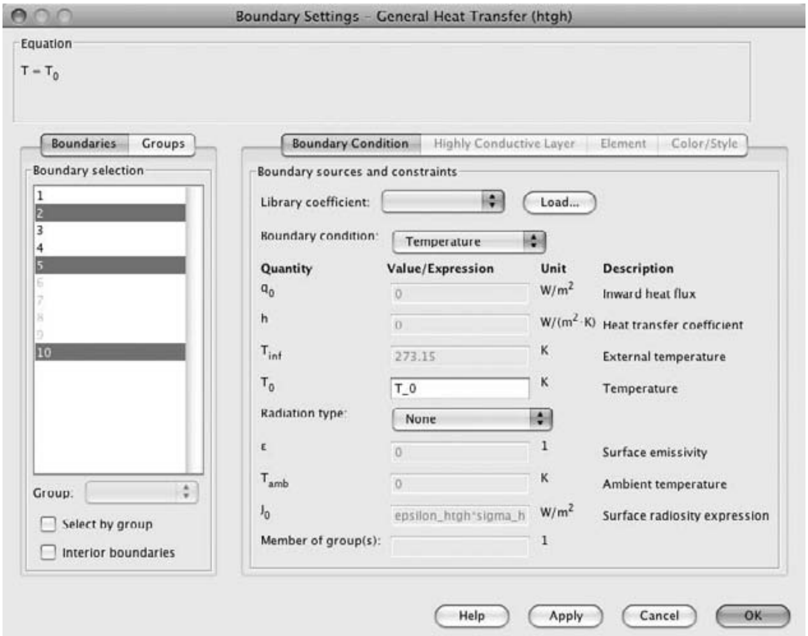


FIGURE 5.48 2D axisymmetric Cylinder_Conduction_3 model Boundary Settings (2, 5, 10) edit window

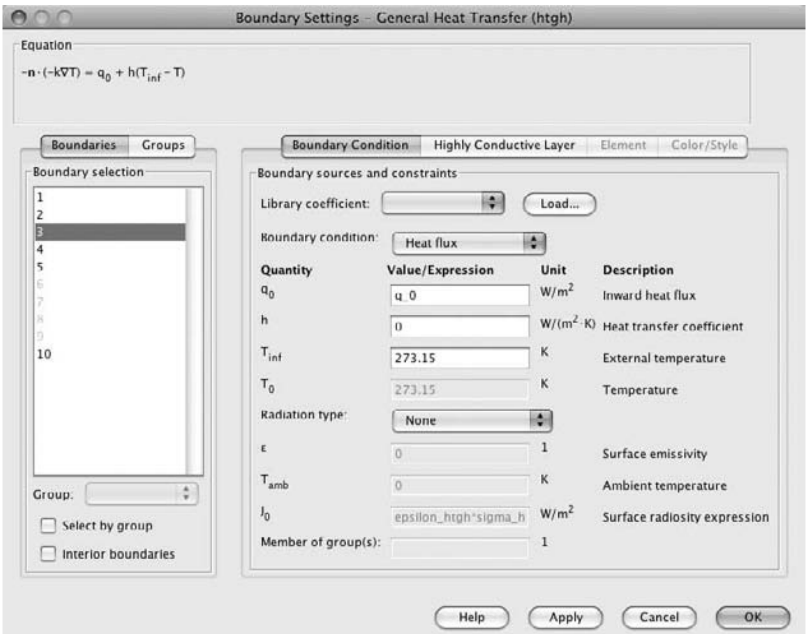


FIGURE 5.49 2D axisymmetric Cylinder_Conduction_3 model Boundary Settings (3) edit window

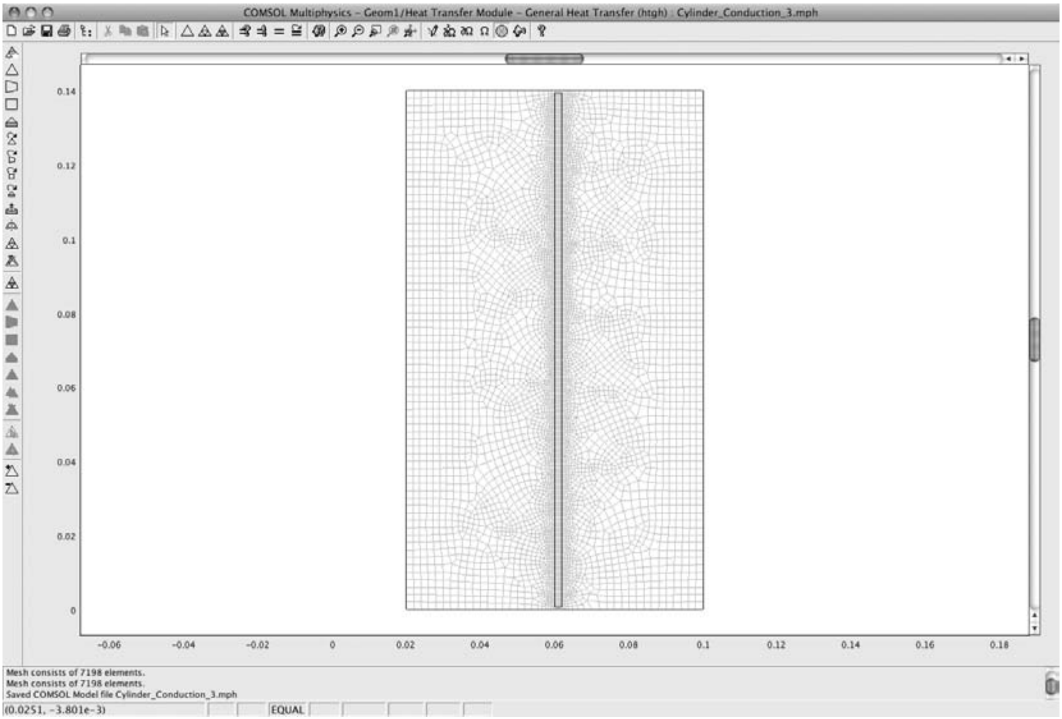


FIGURE 5.50 2D axisymmetric Cylinder_Conduction_3 model mesh

Mesh Generation

From the menu bar, select Mesh > Free Mesh Parameters > Subdomain 1. Enter Maximum element size 0.002. Select Method > Quad. Click the Remesh button. Select Free Mesh Parameters > Subdomain 2. Enter Maximum element size 0.0005. Select Method > Quad. Click the Remesh button.

Click OK. See Figure 5.50.

Solving the 2D Axisymmetric Cylinder_Conduction_3 Model

From the menu bar, select Solve > Solver Parameters > Parametric. Enter q_0 in the Parameter name edit window. Enter 0:5e3:5e5 in the Parameter values edit window. See Figure 5.51. Click OK.

From the menu bar, select Solve > Solve Problem. See Figure 5.52.

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.53.

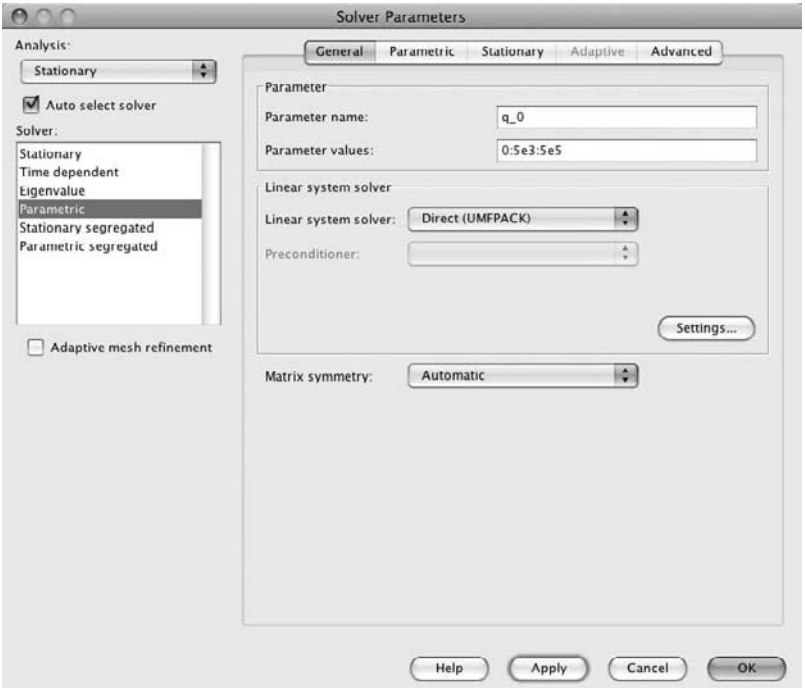


FIGURE 5.51 2D axisymmetric Cylinder_Conduction_3 model Solver Parameters edit window

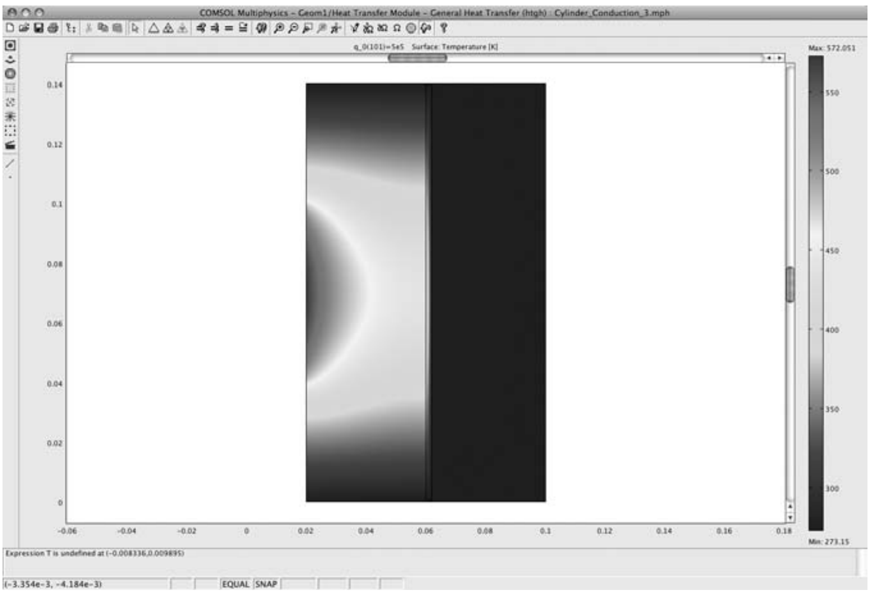


FIGURE 5.52 2D axisymmetric Cylinder_Conduction_3 model using the Parametric Solver (UMFPACK)

FIGURE 5.53 2D axisymmetric Cylinder_Conduction_3 model Plot Parameters window

Click the Start Animation button. See Figure 5.54.

Alternatively, you can play the file Movie5_CC_3.avi that was supplied with this book.

2D Axisymmetric Cylinder Conduction Models: Summary and Conclusions

The models presented in this section of Chapter 5 have introduced the following new concepts: two-dimensional axisymmetric modeling (axial symmetry [2D]), cylindrical coordinates, conductive media DC, Heat Transfer Module, heat conduction theory, opaque and transparent thermally conductive materials, and vacuum. Previously introduced concepts employed in these models include triangular mesh, free mesh parameters, subdomain mesh, maximum element size, and quadrilateral mesh (quad).

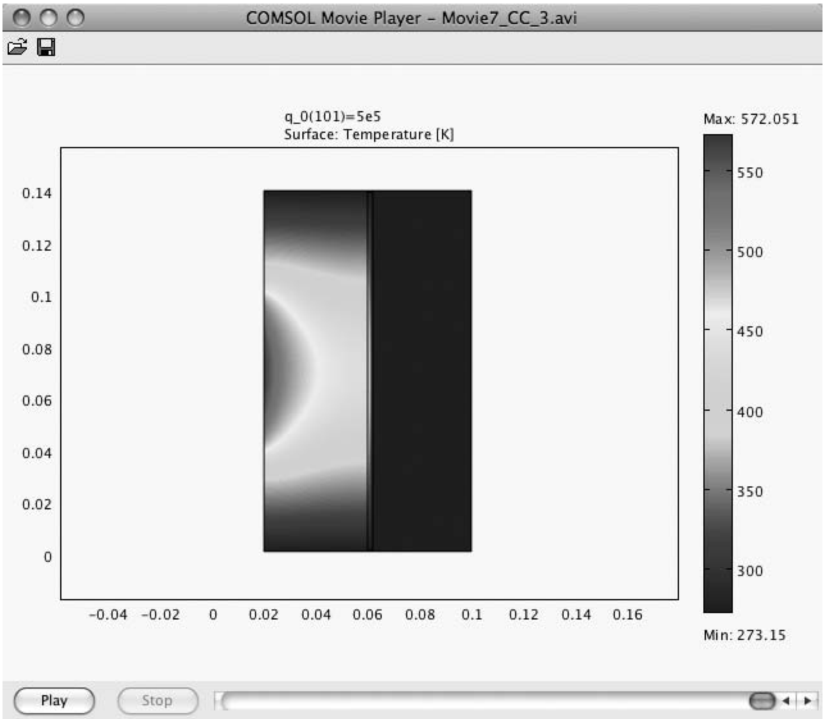


FIGURE 5.54 2D axisymmetric Cylinder_Conduction_3 model animation, final frame

A comparison of the calculated results for the three cylinder conduction models is shown in Table 5.15. As can be readily observed, the presence of a vacuum cavity significantly reduces the rate of heat flow through the model and raises the equilibrium temperature at the surface receiving the heat flux.

For simple heat transfer models, both the basic COMSOL Multiphysics software and the Heat Transfer Module yield the same result, as would be expected. For more complex models involving such conditions as a vacuum, the Heat Transfer Module is required.

Table 5.15 Cylinder Conduction Modeling Results Summary

Model Number	Module Used	Vacuum	T-max (K)	T-max (°C)	$\triangle T$
1p	Heat Transfer Module	No	476.907 K	203.76 °C	—
2	Basic Heat Transfer	No	476.907 K	203.76 °C	0
3	Heat Transfer Module	Yes	572.051 K	298.90 °C	95.144

2D Axisymmetric Insulated Container Design

Sir James Dewar[14] invented the vacuum flask in 1892. The vacuum flask enabled him to store low-temperature liquified gases for longer periods of time. Being a research scientist, his primary concern was the study of the liquification process for gases and the study of the resulting liquids.

NOTE Prior to the mid-1900s, it was uncommon for research scientists to patent or commercialize new inventions, regardless of their potential commercial or economic impact. The usual process was to disclose new findings through letter publication to a learned society.[15]

The term "Thermos"[16] came into existence in 1904, when a German company was formed under the name Thermos GmbH to commercialize the vacuum flask technology. The vacuum flask (invented by Dewar) has come into widespread common usage by both scientists and nonscientists alike, under the name "thermos" or "thermos bottle." As such, the name "thermos" has, through common usage, become the generic name, in the United States and some other countries, for the vacuum flask or thermos bottle. There are, of course, other insulating materials in use that are not quite as efficient as the vacuum flask but nevertheless adequate. Thus some thermos bottles (vacuum flask containers) have no vacuum, but simply a low-thermal-conductivity solid (insulating material) in the place where the vacuum would normally exist.

2D Axisymmetric Thermos_Container Model

NOTE This model is derived from the COMSOL thermos laminar flow and thermos laminar hcoeff models. Those models can be found in the Tutorial Models folder of the Heat Transfer Module Model Library. In this model, Thermos_Container_1, the walls of the flask (e.g., bottle, tank) are formed of stainless steel. In the 2D axisymmetric Thermos_Container_1 model, the selected thermal insulating solid is rigid urethane foam.[17] In the first variation on the 2D axisymmetric Thermos_Container model, a vacuum cavity replaces the urethane foam. In the second variation on the 2D axisymmetric Thermos_Container model, a glass[18] material replaces the stainless steel[19] wall material and the insulating vacuum cavity remains. These changes in the materials design of these models reflect some of the typical alterations and trade-offs that need to be made in the exploratory design phase of a new artifact (e.g., product, tool).

To start building the Thermos_Container_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > General Heat Transfer > Steady-state analysis. See Figure 5.55. Click OK.

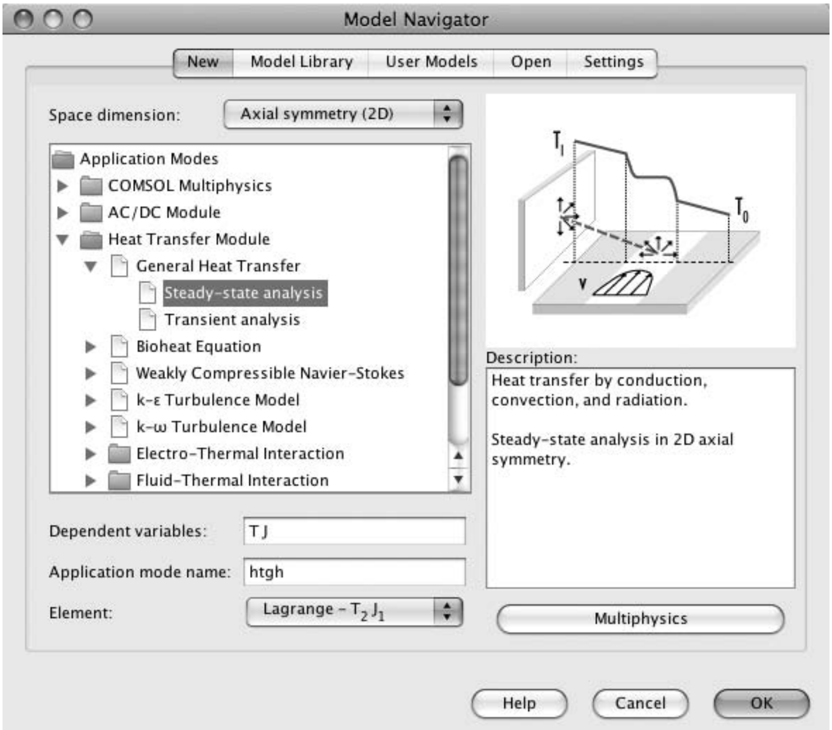


FIGURE 5.55 2D axisymmetric Thermos_Container_1 Model Navigator setup

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.16; also see Figure 5.56. Click OK.

NOTE When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

NOTE When building a model, it is usually best to choose names for modeler-defined parameters (e.g., constants, scalar functions) that are easily recalled and associated with the function that they provide in the model (e.g., k_foam, rho_foam).

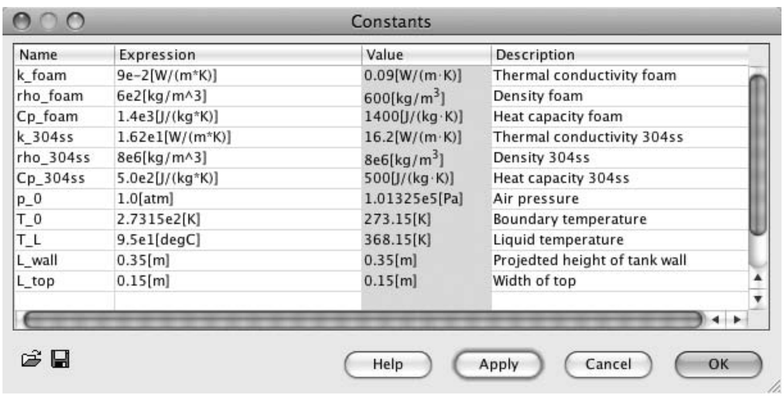
Table 5.16 Constants Edit Window

Name	Expression	Description
k_foam	9e-2[W/(m*K)]	Thermal conductivity foam
rho_foam	6e2[kg/m^3]	Density foam
Cp_foam	1.4e3[J/(kg*K)]	Heat capacity foam
k_304ss	1.62e1[W/(m*K)]	Thermal conductivity 304ss
rho_304ss	8e6[kg/m^3]	Density 304ss
Cp_304ss	5.0e2[J/(kg*K)]	Heat capacity 304ss
p_0	1.0[atm]	Air pressure
T_0	2.7315e2[K]	Boundary temperature
T_L	9.5e1[degC]	Liquid temperature
L_wall	0.35[m]	Projected height of tank wall
L_top	0.15[m]	Width of top

Building the 2D Axisymmetric Thermos Container

NOTE The actual sequence of steps required in the building of the 2D axisymmetric Thermos_Container_1 model is initially somewhat complex. However, once the model is built, the modeler can use the export and import functions to use the same physical model configuration and explore the influence of different materials and materials properties on the overall design behavior, as shown in the first and second variations of the 2D axisymmetric thermos container model.

Using the menu bar, select Draw > Specify Objects > Rectangle. Create each of the two rectangles indicated in Table 5.17.


Constants

Name	Expression	Value	Description
k_foam	9e-2[W/(m*K)]	0.09[W/(m·K)]	Thermal conductivity foam
rho_foam	6e2[kg/m^3]	600[kg/m^3]	Density foam
Cp_foam	1.4e3[J/(kg*K)]	1400[J/(kg·K)]	Heat capacity foam
k_304ss	1.62e1[W/(m*K)]	16.2[W/(m·K)]	Thermal conductivity 304ss
rho_304ss	8e6[kg/m^3]	8e6[kg/m^3]	Density 304ss
Cp_304ss	5.0e2[J/(kg*K)]	500[J/(kg·K)]	Heat capacity 304ss
p_0	1.0[atm]	1.01325e5[Pa]	Air pressure
T_0	2.7315e2[K]	273.15[K]	Boundary temperature
T_L	9.5e1[degC]	368.15[K]	Liquid temperature
L_wall	0.35[m]	0.35[m]	Projedted height of tank wall
L_top	0.15[m]	0.15[m]	Width of top

FIGURE 5.56 2D axisymmetric Thermos_Container_1 model Constants edit window

Table 5.17 Rectangle Edit Window

Object Number	Width	Height	Base	*r*	*z*
Rectangle 1	0.25	0.55	Corner	0	0
Rectangle 2	0.15	0.3	Corner	0	0

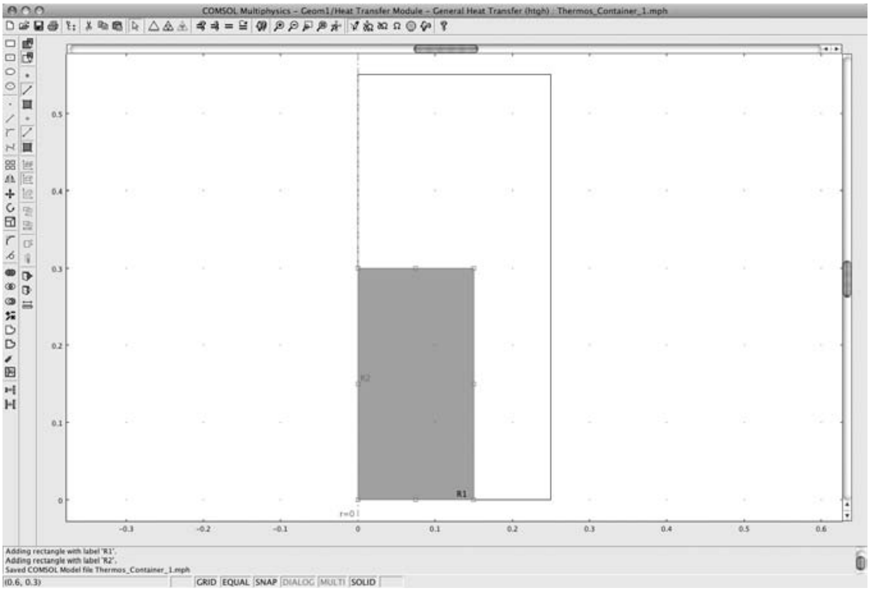


FIGURE 5.57 2D axisymmetric Thermos_Container_1 model rectangles R1 and R2

Click OK, and then click the Zoom Extents button. See Figure 5.57.

Using the menu bar, select Draw > Specify Objects > Ellipse. Create the ellipse indicated in Table 5.18. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Create the rectangle indicated in Table 5.19. Click OK.

Select Draw > Create Composite Object. Enter E1−R3 in the Set formula edit window. Verify that the Keep interior boundaries check box is unchecked. See Figure 5.58. Click OK.

Table 5.18 Ellipse Edit Window

Object Number	A-semiaxes	B-semiaxes	Base	*r*	*z*
Ellipse 1	0.15	0.05	Center	0	0.3

Table 5.19 Rectangle Edit Window

Object Number	Width	Height	Base	*r*	*z*
Rectangle	0.2	0.4	Corner	−0.2	0

Select Draw > Create Composite Object. Enter R2+CO1 in the Set formula edit window. Verify that the Keep interior boundaries check box is unchecked. See Figure 5.59.

Click OK. See Figure 5.60, which shows the profile of the outer tank.

Create the inner structure of the insulated tank by following the steps in Table 5.20. Select the appropriate action from the menu bar using the Draw pull-down menu. See Figure 5.61.

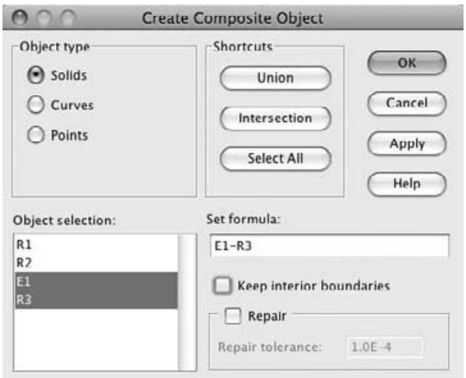


FIGURE 5.58 2D axisymmetric Thermos_Container_1 model half-ellipse creation

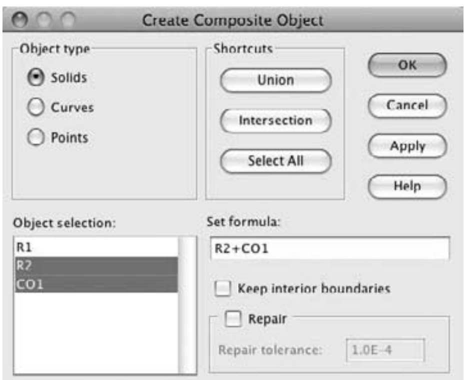


FIGURE 5.59 2D axisymmetric Thermos_Container_1 model outer tank profile creation

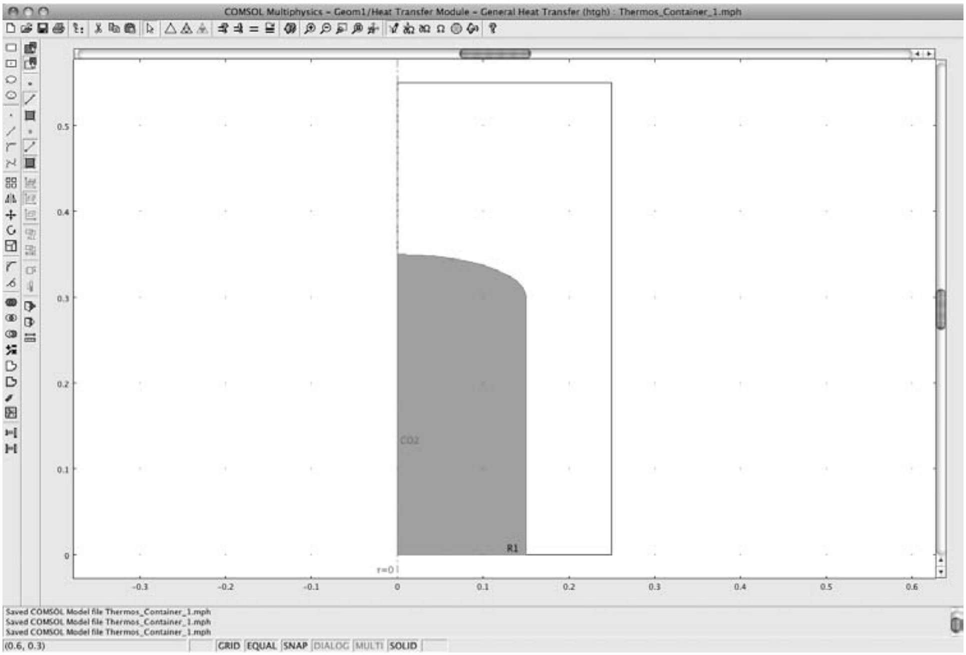


FIGURE 5.60 2D axisymmetric Thermos_Container_1 model outer tank profile

Table 5.20 Tank Structure Creation Steps

Step	Object	Width/A	Height/B	Base	*r*	*z*
1	Rectangle	0.15–0.005	0.3–0.005	Corner	0	0.005
2	Ellipse	0.15–0.005	0.05–0.005	Center	0	0.3
3	Rectangle	0.2	0.4	Corner	−0.2	0
4	Create Composite Object		Formula = E1−R3		No Interior Boundaries	
5	Create Composite Object		Formula = R2+CO1		No Interior Boundaries	
6	Rectangle	0.15–0.025	0.3–0.025	Corner	0	0.025
7	Ellipse	0.15–0.025	0.05–0.025	Center	0	0.3
8	Rectangle	0.2	0.4	Corner	−0.2	0
9	Create Composite Object		Formula = E1−R3		No Interior Boundaries	
10	Create Composite Object		Formula = R2+CO1		No Interior Boundaries	
11	Rectangle	0.15–0.03	0.3–0.03	Corner	0	0.03
12	Ellipse	0.15–0.03	0.05–0.03	Center	0	0.3
13	Rectangle	0.2	0.4	Corner	−0.2	0
14	Create Composite Object		Formula = E1−R3		No Interior Boundaries	
15	Create Composite Object		Formula = R2+CO1		No Interior Boundaries	
16	Rectangle	0.15–0.03	0.29–0.03	Corner	0	0.03

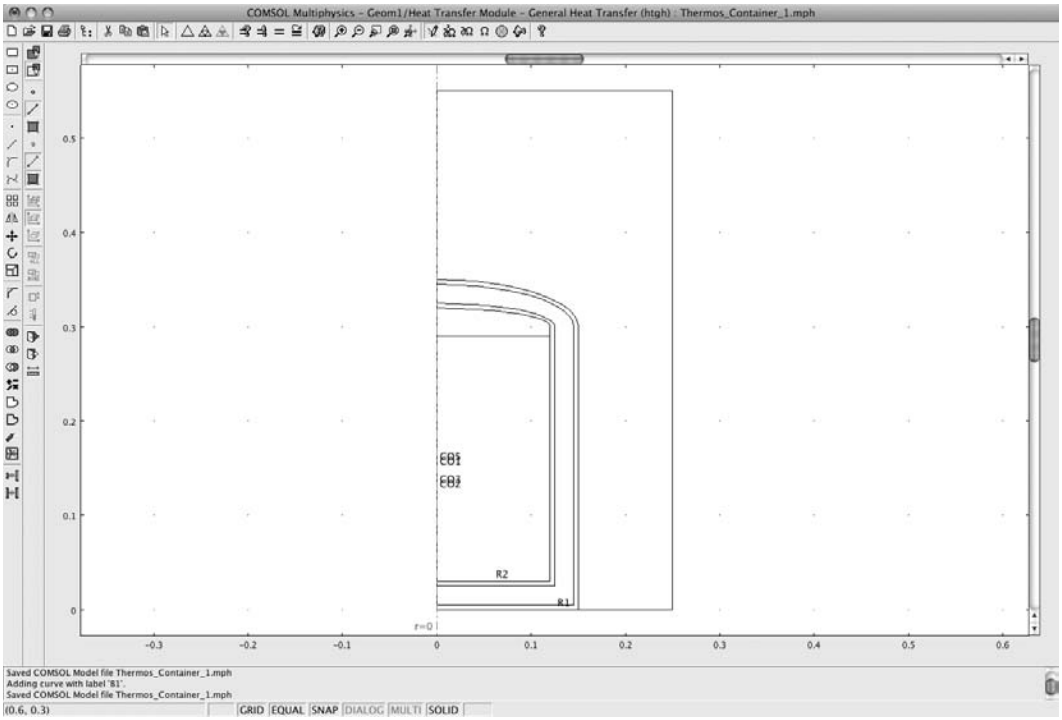


FIGURE 5.61 2D axisymmetric Thermos_Container_1 model tank components

The tank lid structure is defined by adding a line that separates the lid and the body of the tank. Select Draw > Specify Objects > Line. Enter 0.15−0.03 space 0.15 in the r edit window. Enter 0.3 space 0.3 in the z edit window. See Figures 5.62 and 5.63.

The next step is to combine the components into the final tank structure. Select Draw > Create Composite Object. Enter R1+CO2+CO3+CO4+CO5+R2. Important: This time, verify that the Keep interior boundaries check box is *checked*. See Figure 5.64.

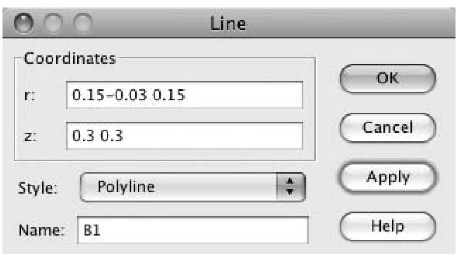


FIGURE 5.62 2D axisymmetric Thermos_Container_1 model Line edit window

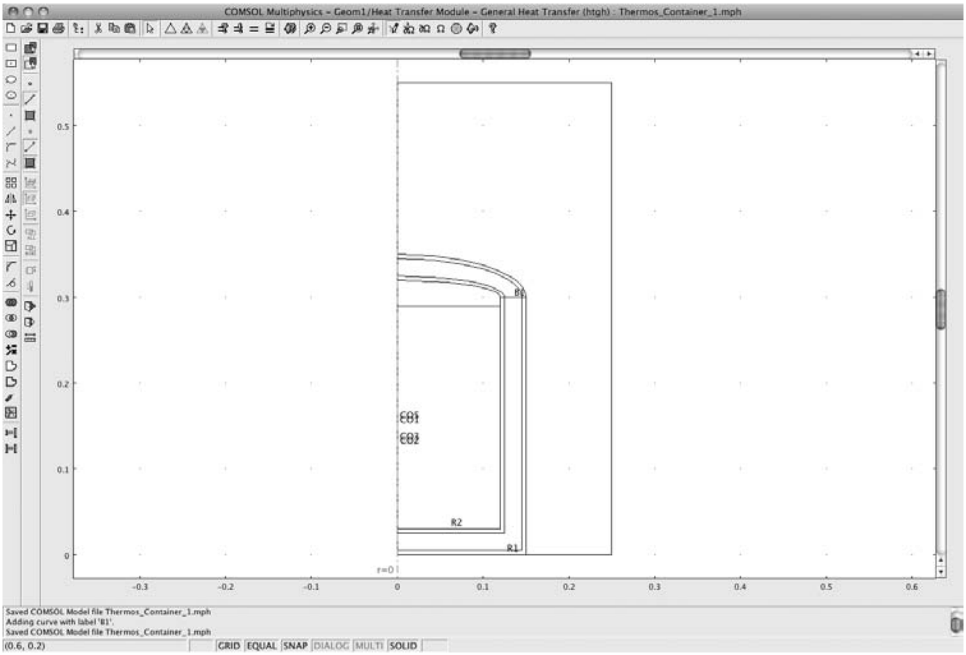


FIGURE 5.63 2D axisymmetric Thermos_Container_1 model tank components with lid line

Click OK. See Figure 5.65.

Now, to save time and effort on the next model, save the present insulated tank configuration. Select File > Export > Geometry Objects to File. Enter TC_1_Geometry in the Save As edit window. Select DXF file (*.dxf) from the File Format pull-down list. See Figure 5.66. Click Save.

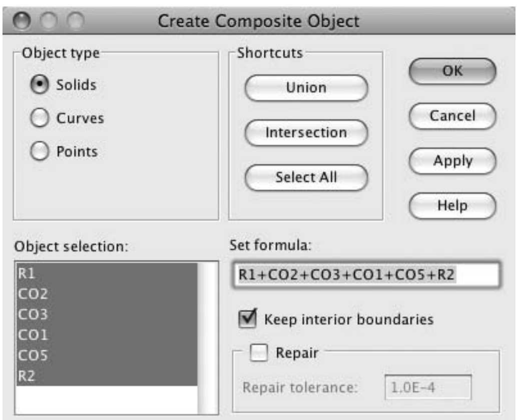


FIGURE 5.64 2D axisymmetric Thermos_Container_1 model Create Composite Object edit window

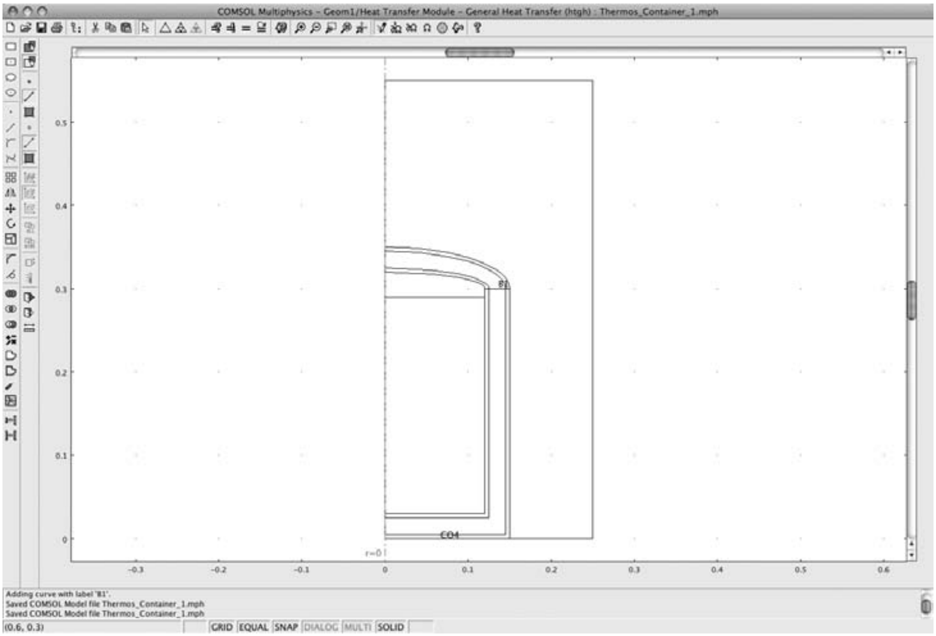


FIGURE 5.65 2D axisymmetric Thermos_Container_1 model tank

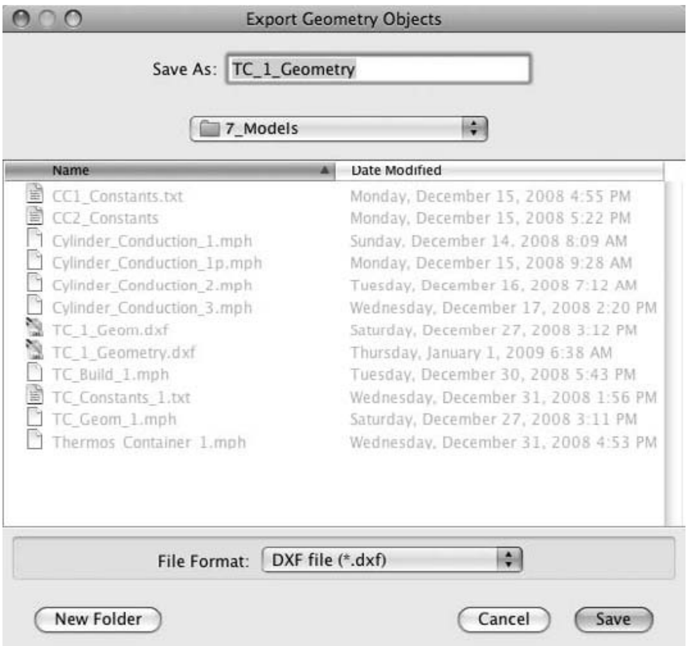


FIGURE 5.66 2D axisymmetric Thermos_Container_1 model tank Export Geometry Objects, Save As window

Table 5.21 Subdomain Edit Window (1, 3, 6, 8)

Subdomain	Operation	Name	Expression	Description
1, 3, 6, 8	Enter	k	k_304ss	Thermal conductivity
1, 3, 6, 8	Enter	ρ	rho_304ss	Density
1, 3, 6, 8	Enter	C_P	Cp_304ss	Heat capacity

Table 5.22 Subdomain Edit Window (2, 7)

Subdomain	Operation	Name	Expression	Description
2, 7	Enter	k	k_foam	Thermal conductivity
2, 7	Enter	ρ	rho_foam	Density
2, 7	Enter	C_P	Cp_foam	Heat capacity

Table 5.23 Subdomain Edit Window (5, 9)

Subdomain	Operation	Name
5, 9	Load	Basic Materials Properties > Air, 1 atm

Table 5.24 Subdomain Edit Window (4)

Subdomain	Operation	Name
4	Load	Basic Materials Properties > Water, liquid

Physics Subdomain Settings: General Heat Transfer

Having established the geometry for the 2D axisymmetric Thermos_Container_1 model, the next step is to define the fundamental Physics conditions. In the Subdomain edit windows, load or enter the information shown in Tables 5.21 through 5.24. See also corresponding Figures 5.67 through 5.70.

Enter p_0 in place of p in the density expression to yield rho(p_0[1/Pa],T[1/K])[kg/m^3] in the Density edit window.

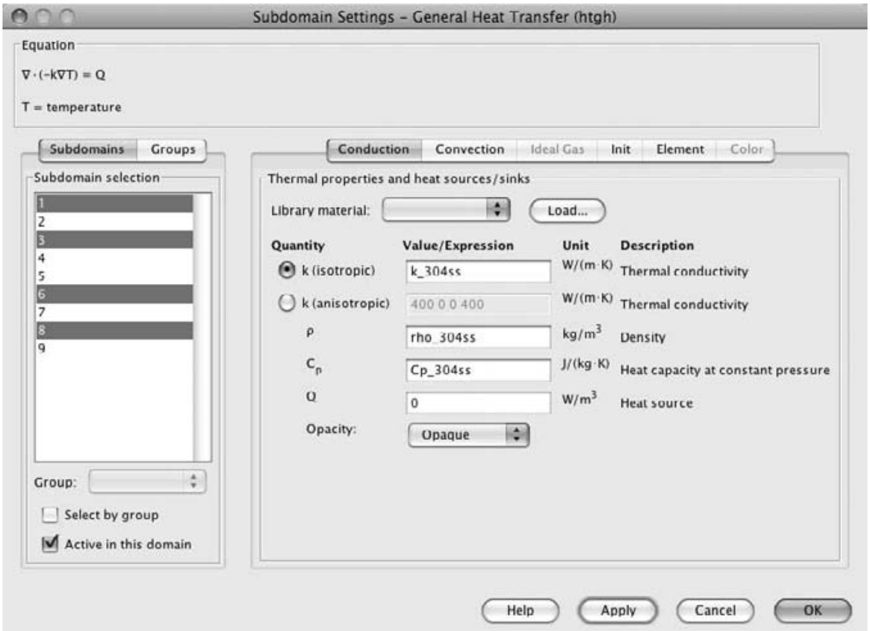


FIGURE 5.67 2D axisymmetric Thermos_Container_1 model Subdomain Settings (1, 3, 6, 8) edit window

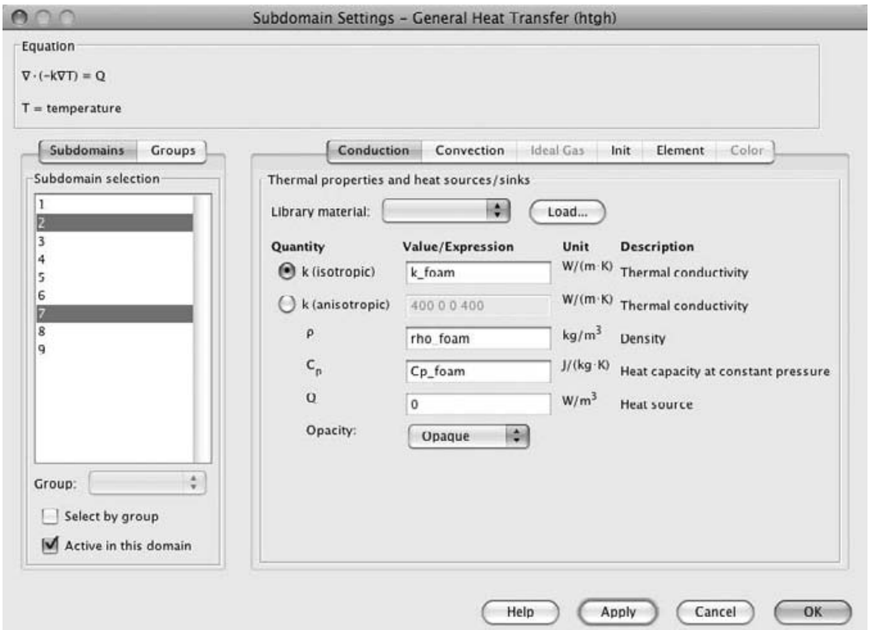


FIGURE 5.68 2D axisymmetric Thermos_Container_1 model Subdomain Settings (2, 7) edit window

FIGURE 5.69 2D axisymmetric Thermos_Container_1 model Subdomain Settings (5, 9) edit window

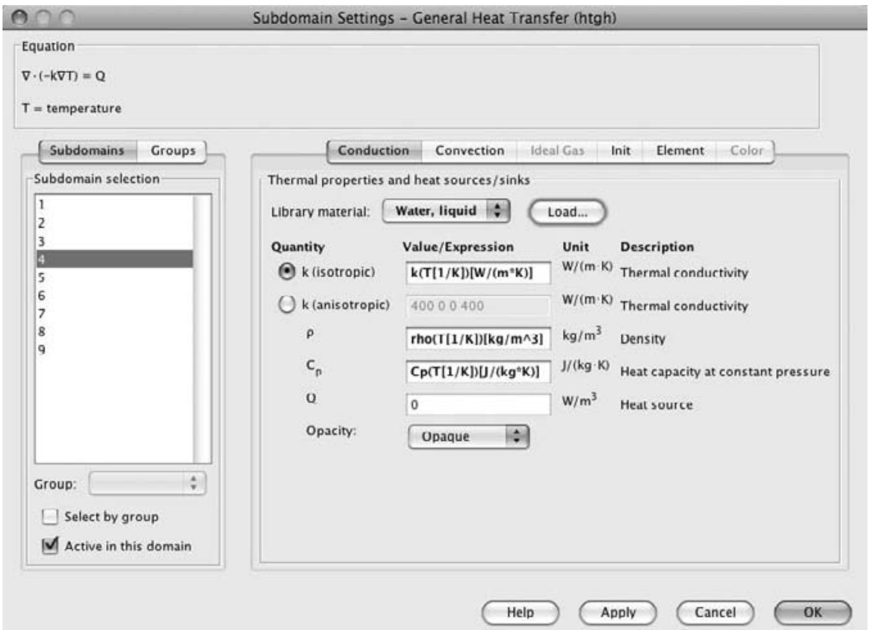


FIGURE 5.70 2D axisymmetric Thermos_Container_1 model Subdomain Settings (4) edit window

Table 5.25 Boundary Settings–General Heat Transfer Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
2	Thermal insulation	—	5.71
8, 14, 21	Temperature	T_L	5.72

NOTE In this model, because of the high thermal conductivity of water, the temperature of the liquid is very close to uniform throughout. Thus uniformity can be assumed and subdomain 4 can be made inactive. The temperature of the liquid is incorporated into the boundary conditions. Also, because this model will calculate only the heat transfer, and not the detailed convection flow in the surrounding air, subdomain 9 can be made inactive. Convection losses are incorporated into the heat transfer coefficient boundary conditions. Incorporating both of these assumptions into the model significantly simplifies the model calculations.

Select subdomain 4. Uncheck the Active in this domain check box.

Select subdomain 9. Uncheck the Active in this domain check box. Click OK.

Physics Boundary Settings: General Heat Transfer

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 5.25. Click OK. See Figures 5.71 and 5.72.

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select the boundary condition.

Load the Library coefficient using the path Heat Transfer Coefficients > Air, Ext. Natural Convection as shown in Table 5.26.

Enter L_wall in the Heat transfer coefficient (h) expression in place of the L_htgh term for boundary 28:

h_ave(T[1/K],Tinf_htgh[1/K],L_wall[1/m])[W/(m^2*K)]

Enter T_0 in the External temperature (T_{inf}) edit window for boundary 28. See Figure 5.73.

Table 5.26 Boundary Settings–General Heat Transfer Edit Window

Boundary	Boundary Condition	Library Coefficient	Figure Number
28	Heat flux	Nat. Vertical wall, L=height	5.73
34	Heat flux	Nat. Horiz. plane, Upside, L=width	5.74

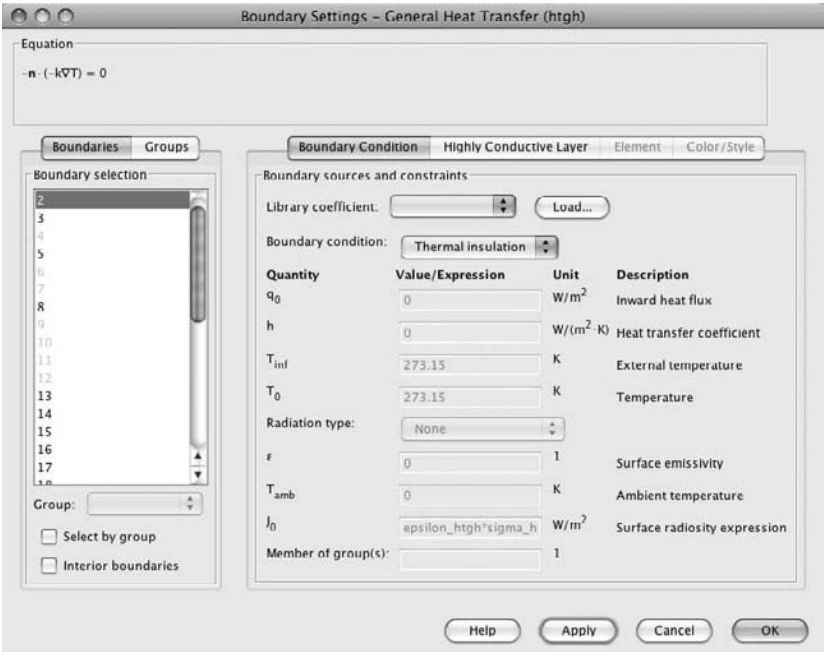


FIGURE 5.71 2D axisymmetric Thermos_Container_1 model Boundary Settings (2) edit window

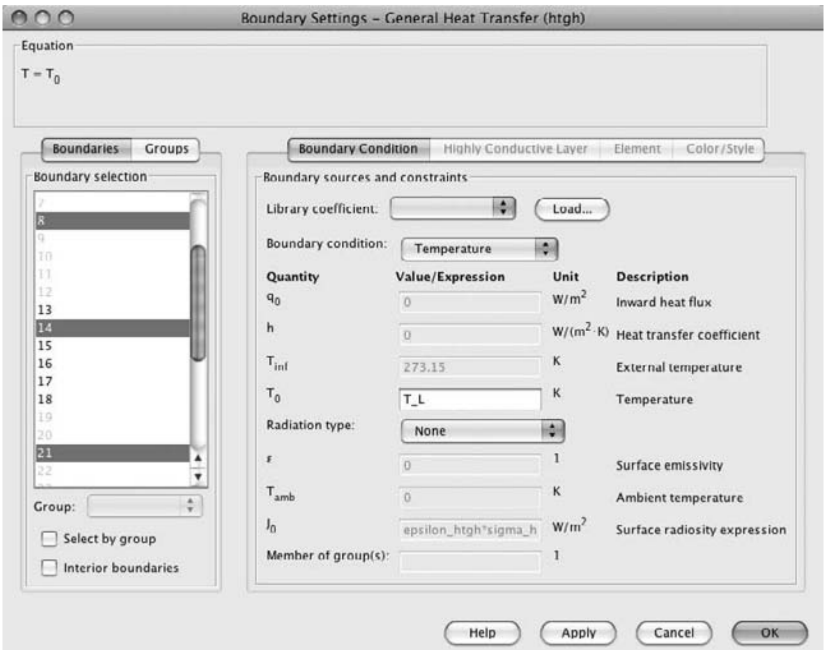


FIGURE 5.72 2D axisymmetric Thermos_Container_1 model Boundary Settings (8, 14, 21) edit window

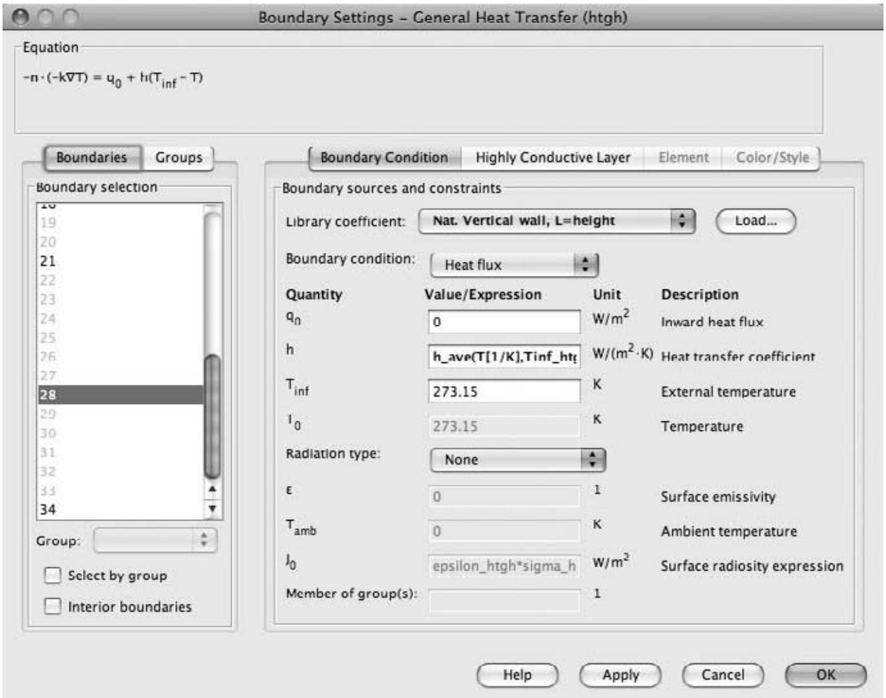


FIGURE 5.73 2D axisymmetric Thermos_Container_1 model Boundary Settings (28) edit window

Enter L_top in the Heat transfer coefficient (h) expression in place of the L_htgh term for boundary 34:

h_ave(T[1/K],Tinf_htgh[1/K],L_top[1/m])[W/(m^2*K)]

Enter T_0 in the External temperature (T_{inf}) edit window for boundary 34. See Figure 5.74.

Click OK.

Mesh Generation

From the menu bar, select Mesh > Free Mesh Parameters > Subdomain 1, 2, 3, 6, 7, 8. Enter Maximum element size 0.002. Select Method > Quad. Click the Remesh button. See Figure 5.75.

Click OK. See Figure 5.76.

Solving the 2D Axisymmetric Thermos_Container_1 Model

From the menu bar, select Solve > Solver Parameters > Parametric. Enter T_L in the Parameter name edit window. Enter 273.15:9.5:368.15 in the Parameter values edit window. See Figure 5.77. Click OK.

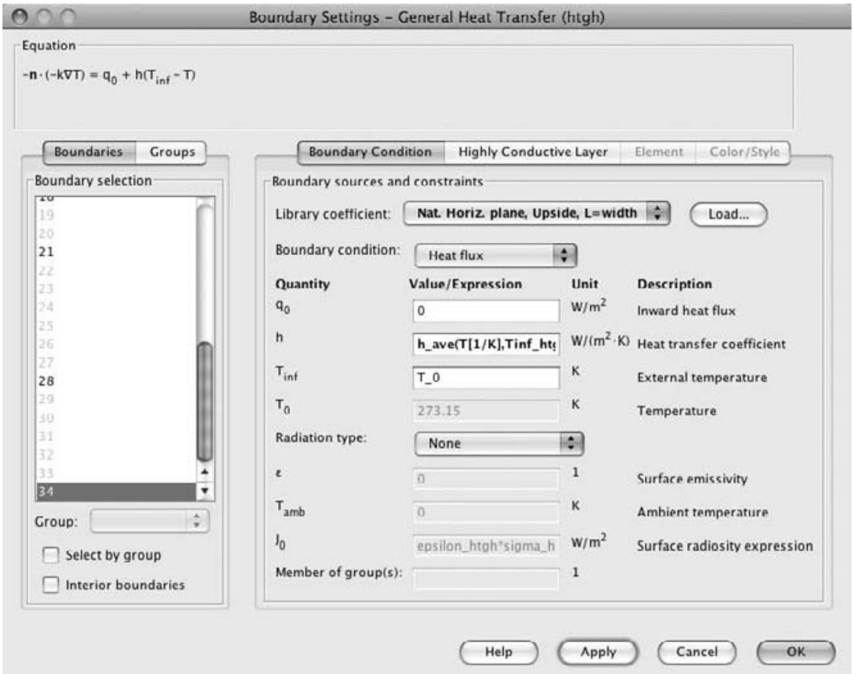


FIGURE 5.74 2D axisymmetric Thermos_Container_1 model Boundary Settings (34) edit window

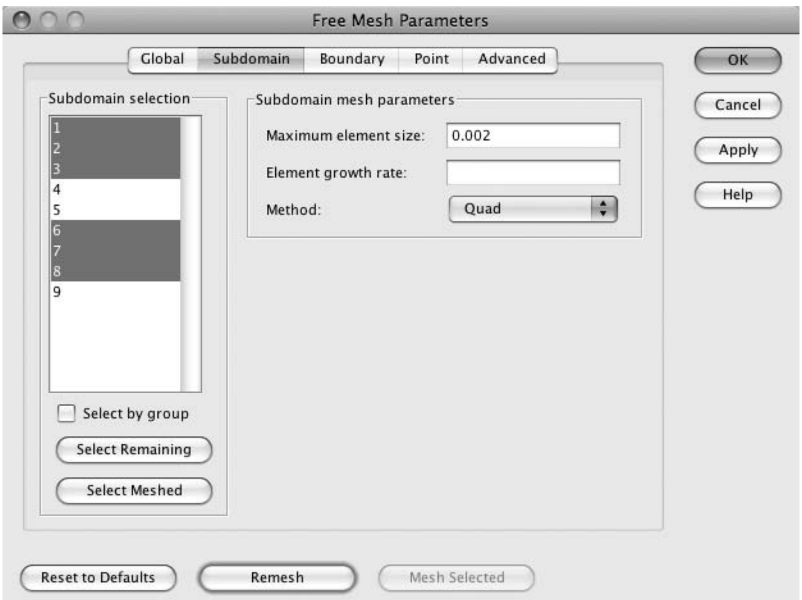


FIGURE 5.75 2D axisymmetric Thermos_Container_1 model Free Mesh Parameters edit window

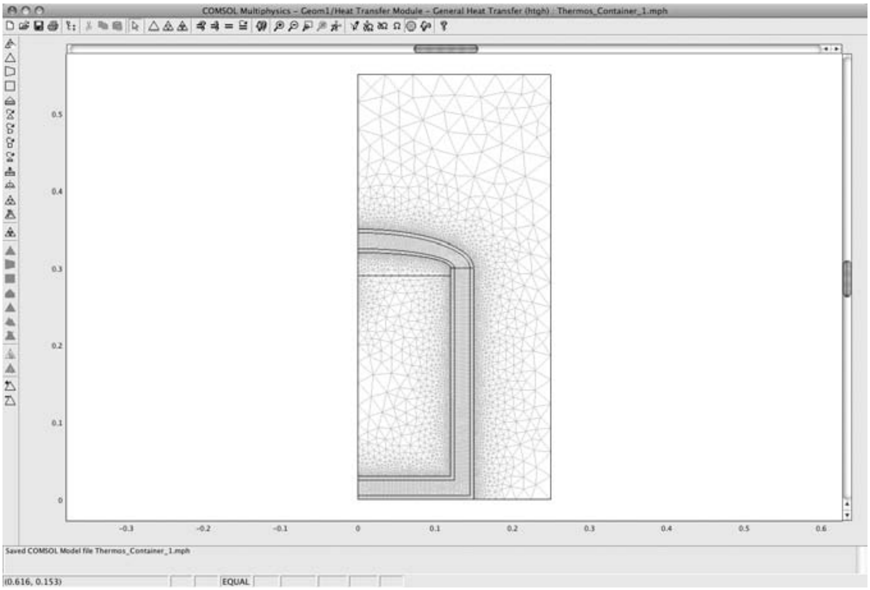


FIGURE 5.76 2D axisymmetric Thermos_Container_1 model mesh

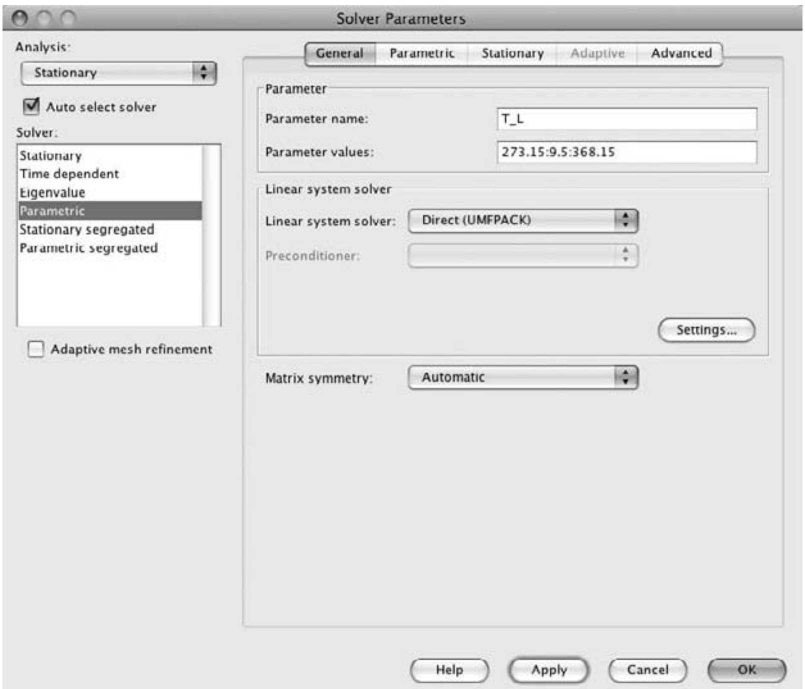


FIGURE 5.77 2D axisymmetric Thermos_Container_1 model Solver Parameters edit window

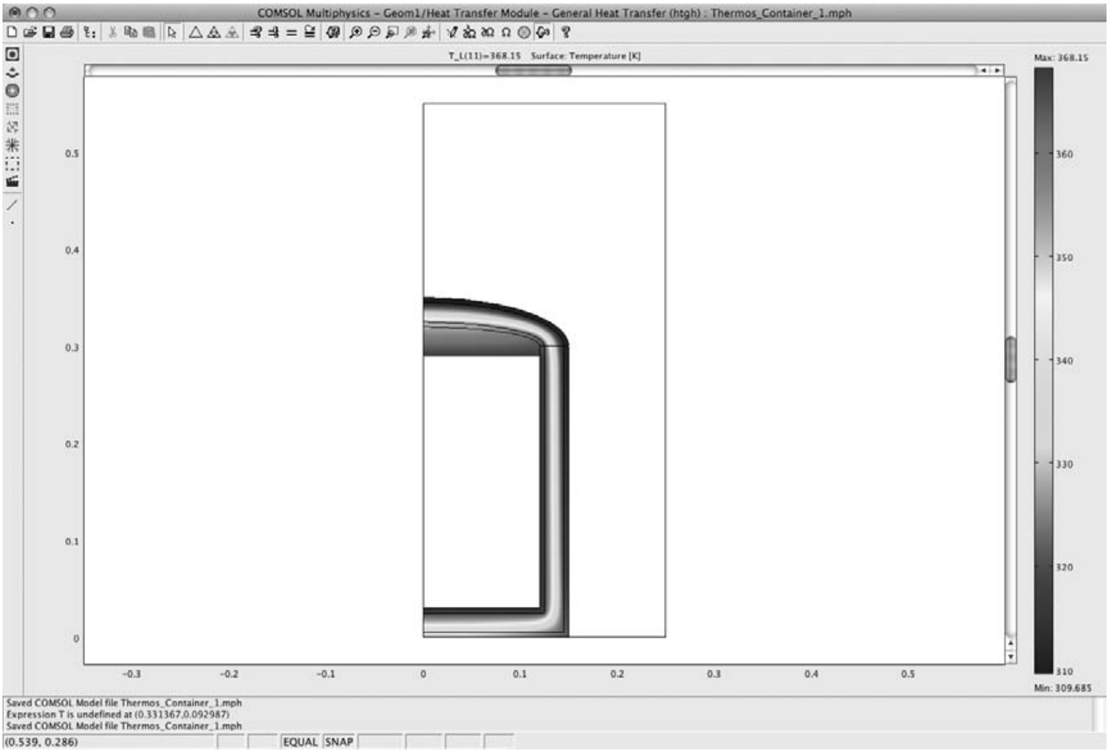


FIGURE 5.78 2D axisymmetric Thermos_Container_1 model using the Parametric Solver (UMFPACK)

From the menu bar, select Solve > Solve Problem. See Figure 5.78.

Postprocessing

Select Postprocessing > Plot Parameters > Surface. Select "°C [degC]" in the Unit pull-down list. See Figure 5.79.

Click OK. See Figure 5.80.

Given that our main interest in creating the 2D axisymmetric Thermos_Container_1 model is to examine the heat transfer, the next step is to determine the heat loss. Select Postprocessing > Boundary Integration. Select boundaries 28 and 34 (the wall and top of the tank, respectively). Select "Normal total heat flux" in the Predefined quantities pull-down list. Check the Compute surface integral (for axisymmetric modes) check box. See Figure 5.81.

Click OK. The result of the Boundary Integration (~82 W) is displayed as Value of surface integral: xx.xxxxx [W], Expression: ntflux_htgh, Boundaries: 28, 34 in the display window at the bottom of the COMSOL user interface. See Figure 5.82.

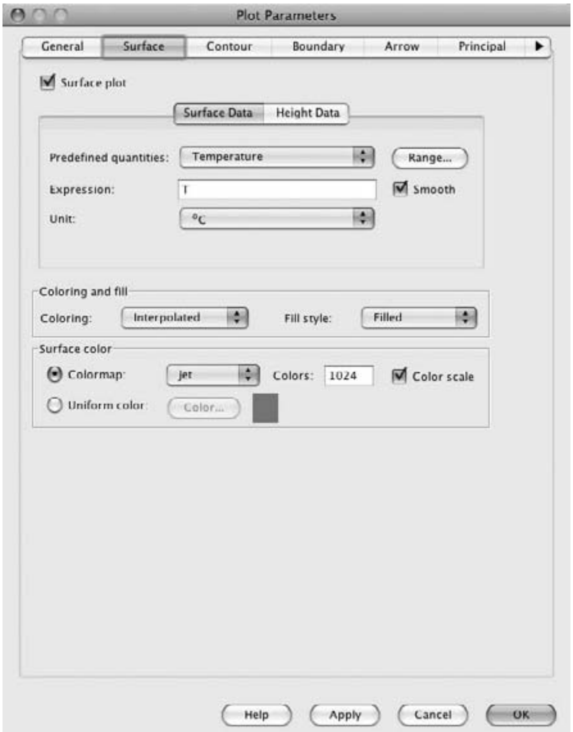


FIGURE 5.79 2D axisymmetric Thermos_Container_1 model Plot Parameters window

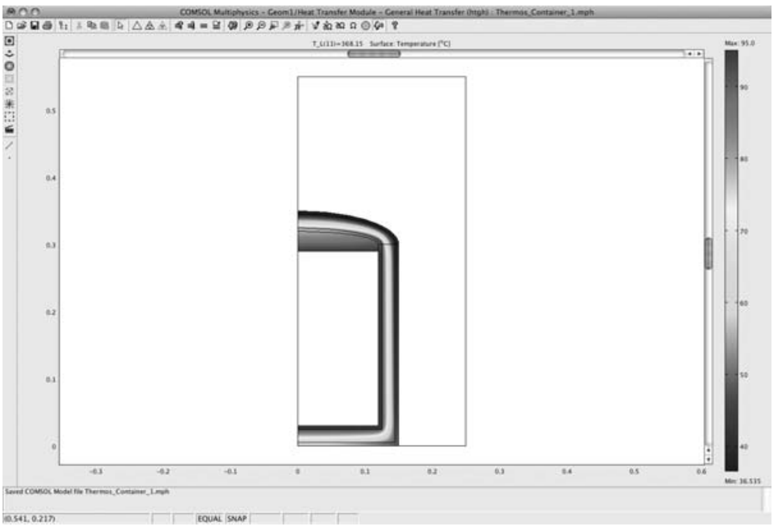


FIGURE 5.80 2D axisymmetric Thermos_Container_1 model surface temperature (°C)

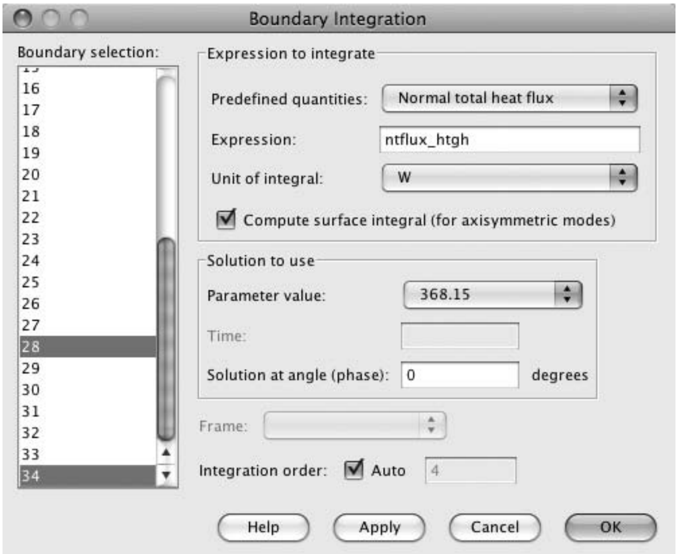


FIGURE 5.81 2D axisymmetric Thermos_Container_1 model Boundary Integration edit window

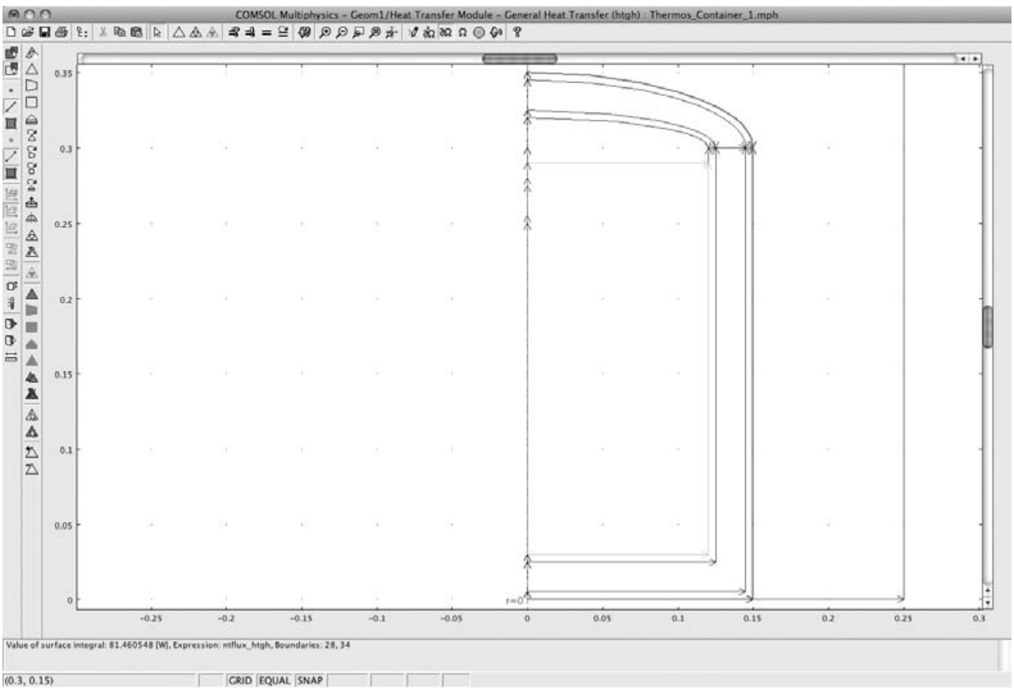


FIGURE 5.82 2D axisymmetric Thermos_Container_1 model user interface display window

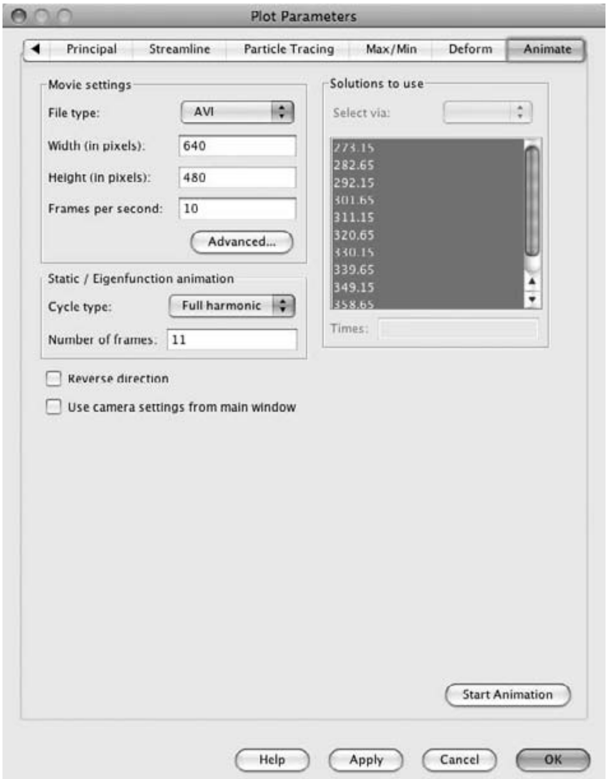


FIGURE 5.83 2D axisymmetric Thermos_Container_1 model Plot Parameters window

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.83.

Click the Start Animation button. See Figure 5.84.

Alternatively, you can play the file Movie5_TC_1.avi that was supplied with this book.

First Variation on the 2D Axisymmetric Thermos_Container Model

NOTE In this model, Thermos_Container_2, the walls of the flask (e.g., bottle, tank) are formed of stainless steel. In the 2D axisymmetric Thermos_Container_1 model, the selected thermal insulating solid was rigid urethane foam. In this model, the first variation on the 2D axisymmetric Thermos_Container model, a vacuum cavity replaces the urethane foam.

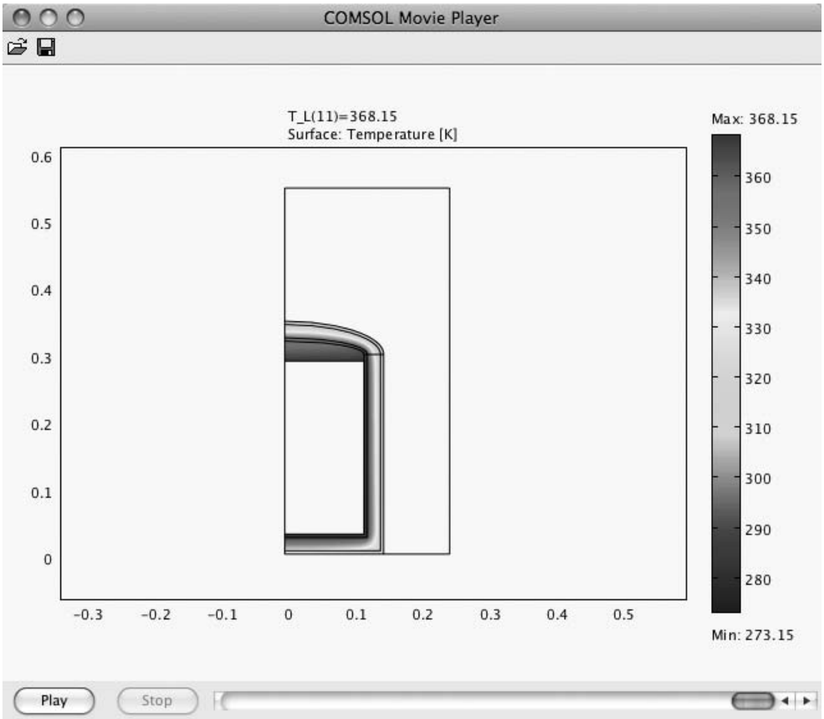


FIGURE 5.84 2D axisymmetric Thermos_Container_1 model animation, final frame

To start building the Thermos_Container_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > General Heat Transfer > Steady-state analysis. See Figure 5.85. Click OK.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.27; also see Figure 5.86. Click OK.

NOTE When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

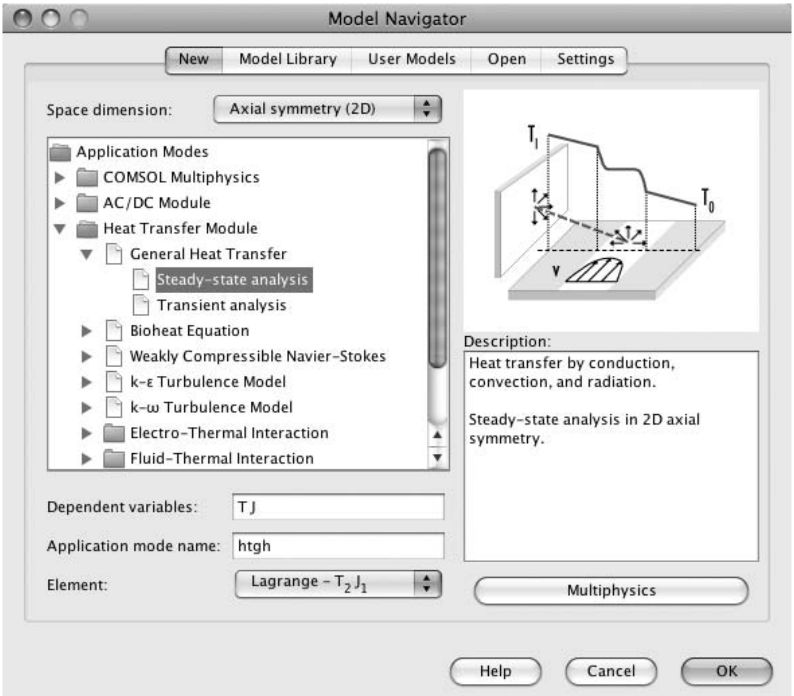
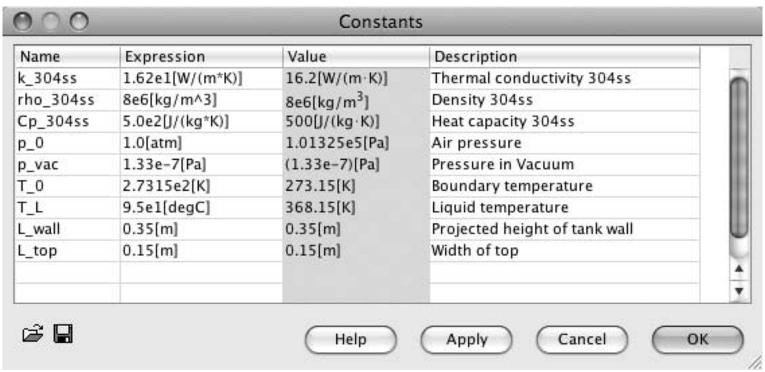


FIGURE 5.85 2D axisymmetric Thermos_Container_2 Model Navigator setup

---

NOTE When building a model, it is usually best to choose a name for modeler-defined parameters (e.g., constants, scalar functions) that are easily recalled and associated with the function that they provide in the model (e.g., p_vac, L_wall).


Constants

Name	Expression	Value	Description
k_304ss	1.62e1[W/(m*K)]	16.2[W/(m·K)]	Thermal conductivity 304ss
rho_304ss	8e6[kg/m^3]	8e6[kg/m^3]	Density 304ss
Cp_304ss	5.0e2[J/(kg*K)]	500[J/(kg·K)]	Heat capacity 304ss
p_0	1.0[atm]	1.01325e5[Pa]	Air pressure
p_vac	1.33e-7[Pa]	(1.33e-7)[Pa]	Pressure in Vacuum
T_0	2.7315e2[K]	273.15[K]	Boundary temperature
T_L	9.5e1[degC]	368.15[K]	Liquid temperature
L_wall	0.35[m]	0.35[m]	Projected height of tank wall
L_top	0.15[m]	0.15[m]	Width of top

FIGURE 5.86 2D axisymmetric Thermos_Container_2 model Constants edit window

Table 5.27 Constants Edit Window

Name	Expression	Description
k_304ss	1.62e1[W/(m*K)]	Thermal conductivity 304ss
rho_304ss	8e6[kg/m^3]	Density 304ss
Cp_304ss	5.0e2[J/(kg*K)]	Heat capacity 304ss
p_0	1.0[atm]	Air pressure
p_vac	1.33e-7[Pa]	Pressure in vacuum
T_0	2.7315e2[K]	Boundary temperature
T_L	9.5e1[degC]	Liquid temperature
L_wall	0.35[m]	Projected height of tank wall
L_top	0.15[m]	Width of top

Select File > Save as. Enter Thermos_Container_2. Click the Save button.

Importing the 2D Axisymmetric Thermos Container

NOTE The actual sequence of steps required in the building of the 2D axisymmetric Thermos_Container was presented in the discussion of the 2D axisymmetric Thermos_Container_1 model. Now the modeler can use the import function to utilize the same physical model configuration and explore the influence of different materials and materials properties on the overall model design behavior.

Using the menu bar, select File > Import > CAD Data From File. Select "TC_1_Geometry.dxf." See Figure 5.87. Click the Import button.

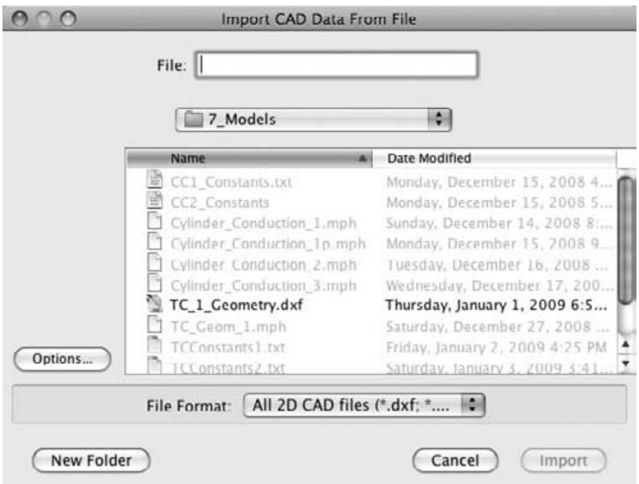


FIGURE 5.87 2D axisymmetric Thermos_Container_2 model import

Table 5.28 Rectangle Edit Window

Object Number	Width	Height	Base	*r*	*z*
Rectangle 1	0.25	0.55	Corner	0	0

NOTE Because the Geometry.dxf file contains only boundary information, the modeler needs to create a domain to which the boundary information can be applied.

Using the menu bar, select Draw > Specify Objects > Rectangle, as indicated in Table 5.28. Click OK. See Figure 5.88.

Physics Subdomain Settings: General Heat Transfer

Having established the geometry for the 2D axisymmetric Thermos_Container_2 model, the next step is to define the fundamental Physics conditions. Select Physics > Subdomain Settings. In the Subdomain edit windows, enter the information shown in Table 5.29. See Figure 5.89.

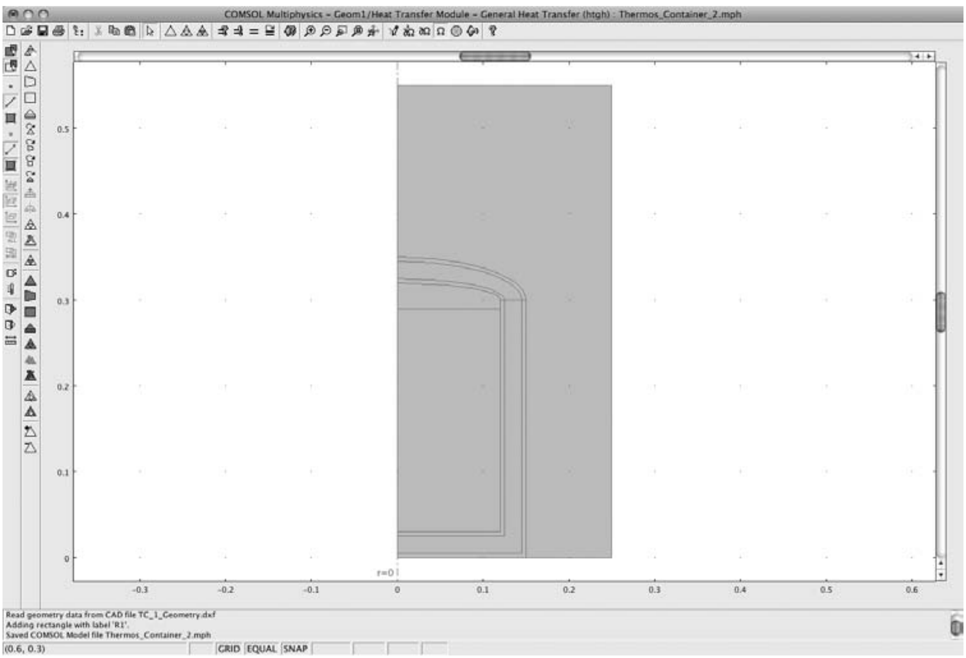


FIGURE 5.88 2D axisymmetric Thermos_Container_2 model import and rectangle R1

Table 5.29 Subdomain Edit Window

Subdomain	Operation	Name	Expression	Description
1, 3, 6, 8	Enter	k	k_304ss	Thermal conductivity
1, 3, 6, 8	Enter	ρ	rho_304ss	Density
1, 3, 6, 8	Enter	C_P	Cp_304ss	Heat capacity

In the Subdomain edit windows, enter the information shown in Table 5.30. Enter p_vac in place of p in the density expression rho(p[1/Pa],T[1/K])[kg/m^3], so that it reads rho(p_vac[1/Pa],T[1/K])[kg/m^3] in the Density edit window. Select "Transparent" from the Opacity pull-down list. See Figure 5.90.

In the Subdomain edit windows, enter the information shown in Table 5.31. Enter p_0 in place of p in the density expression to yield rho(p_0[1/Pa],T[1/K])[kg/m^3] in the Density edit window. Select "Transparent" from the Opacity pull-down list. See Figure 5.91.

In the Subdomain edit windows, enter the information shown in Table 5.32. See Figure 5.92.

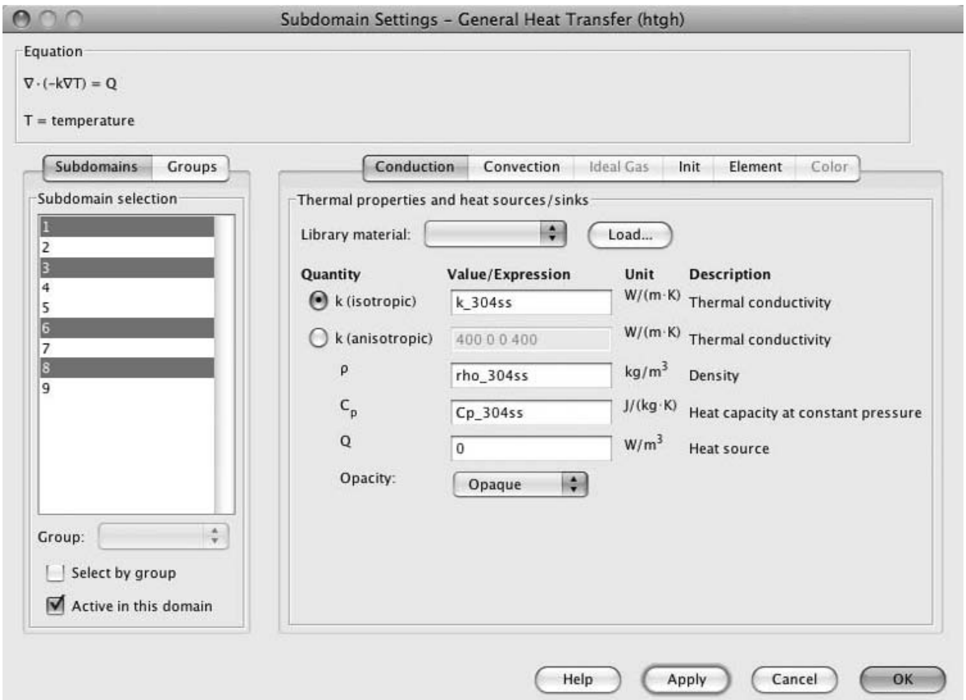


FIGURE 5.89 2D axisymmetric Thermos_Container_2 model Subdomain Settings (1, 3, 6, 8) edit window

Table 5.30 Subdomain Edit Window

Subdomain	Operation	Name
2, 7	Load	Basic Materials Properties > Air, 1 atm

Table 5.31 Subdomain Edit Window

Subdomain	Operation	Name
5, 9	Load	Basic Materials Properties > Air, 1 atm

Table 5.32 Subdomain Edit Window

Subdomain	Operation	Name
4	Load	Basic Materials Properties > Water, liquid

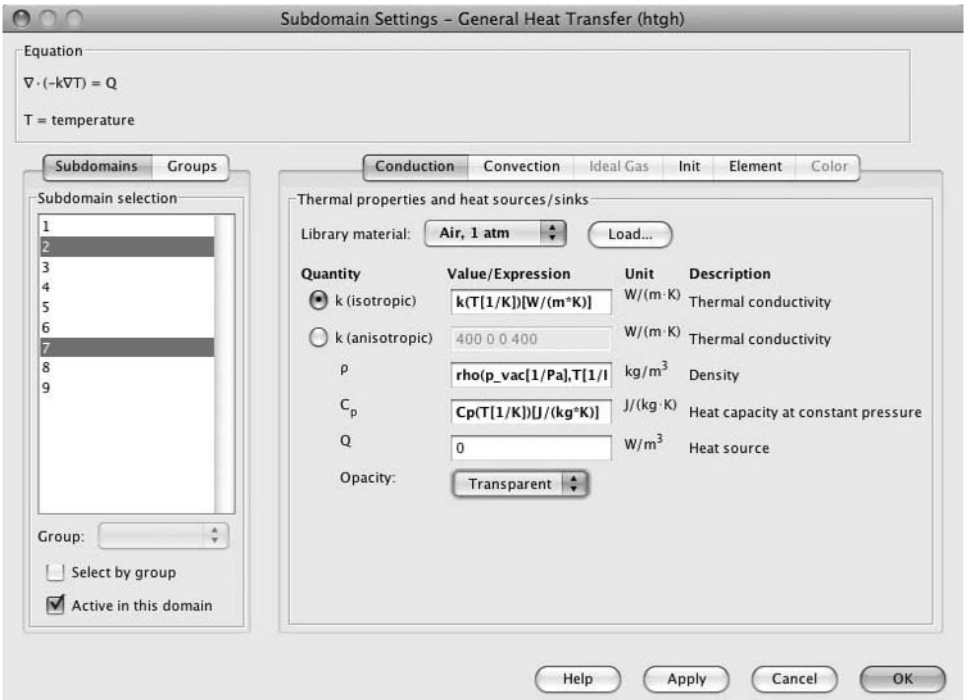


FIGURE 5.90 2D axisymmetric Thermos_Container_2 model Subdomain Settings (2, 7) edit window

FIGURE 5.91 2D axisymmetric Thermos_Container_2 model Subdomain Settings (5, 9) edit window

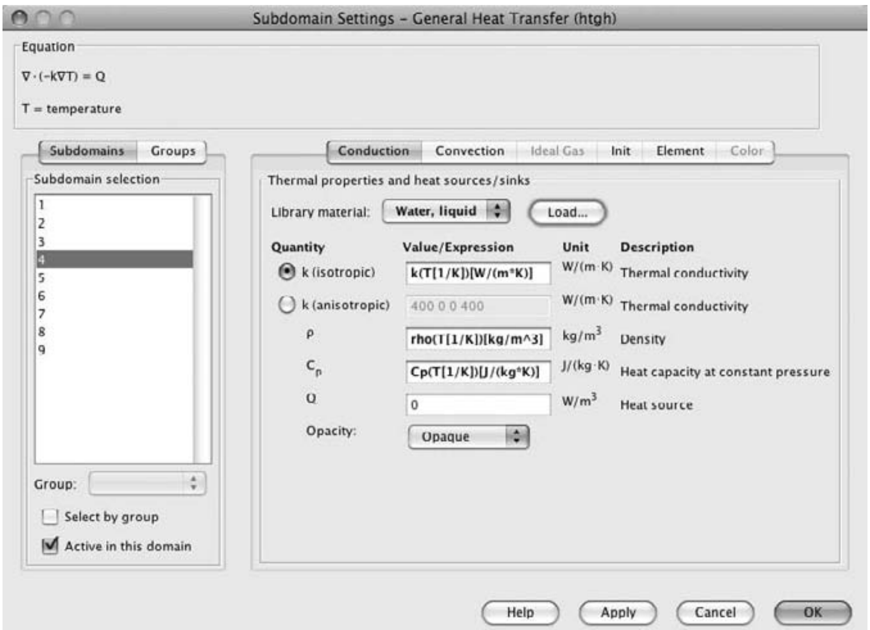


FIGURE 5.92 2D axisymmetric Thermos_Container_2 model Subdomain Settings (4) edit window

Table 5.33 Boundary Settings–General Heat Transfer Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
2	Thermal insulation	—	5.93
8, 14, 21	Temperature	T_L	5.94

NOTE In this model, because of the high thermal conductivity of water, the temperature of the liquid is very close to uniform throughout. Thus uniformity can be assumed and subdomain 4 can be made inactive. The temperature of the liquid is incorporated into the boundary conditions. Also, because this model will calculate only the heat transfer, and not the detailed convection flow in the surrounding air, subdomain 9 can be made inactive. Convection losses are incorporated into the heat transfer coefficient boundary conditions. Incorporating both of these assumptions into the model significantly simplifies the model calculations.

Select subdomain 4. Uncheck the Active in this domain check box. Select subdomain 9. Uncheck the Active in this domain check box. Click OK.

Physics Boundary Settings: General Heat Transfer

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 5.33. Click OK. See Figures 5.93 and 5.94.

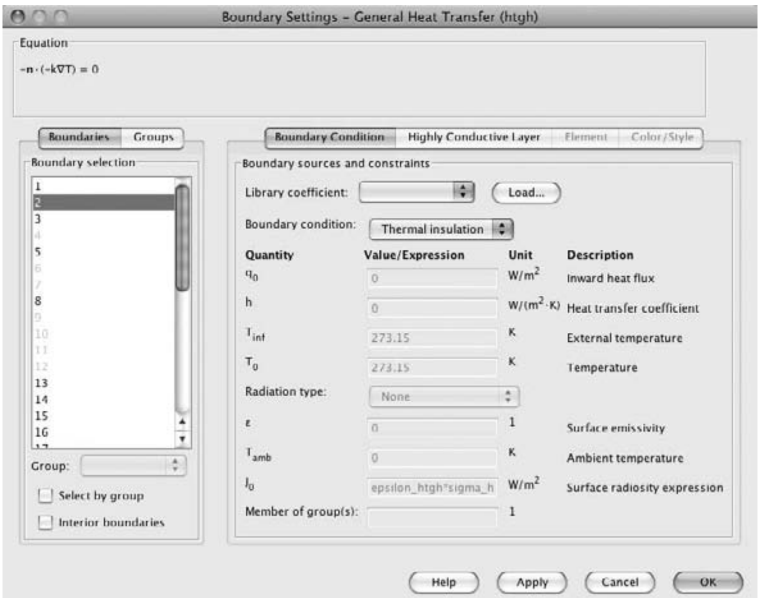


FIGURE 5.93 2D axisymmetric Thermos_Container_2 model Boundary Settings (2) edit window

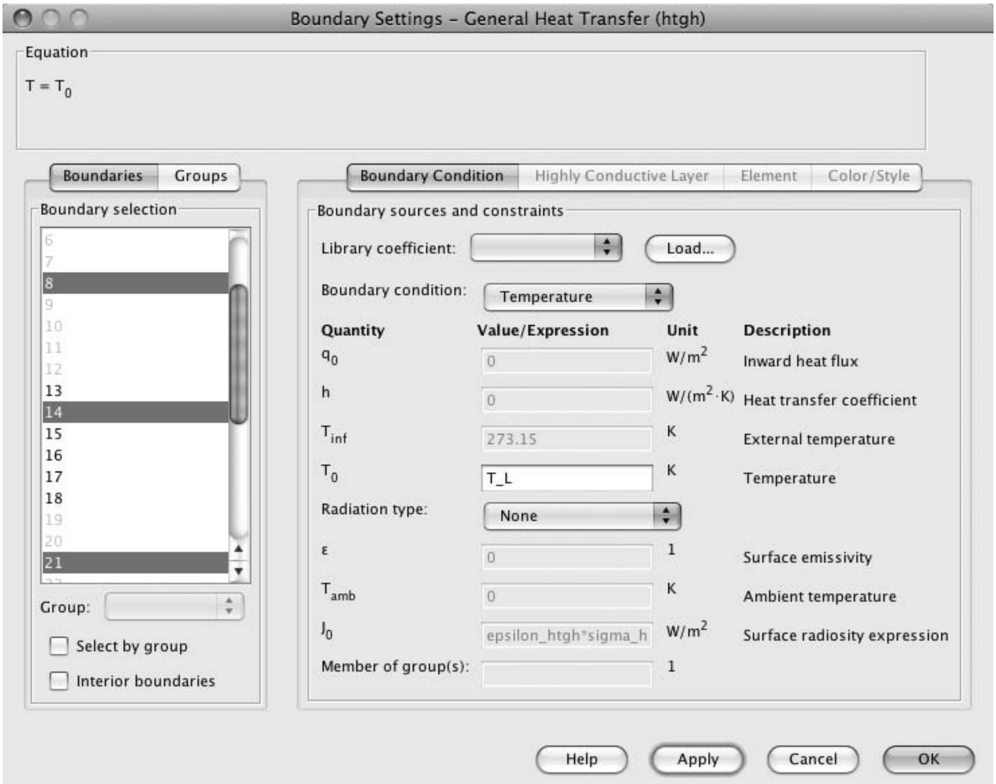


FIGURE 5.94 2D axisymmetric Thermos_Container_2 model Boundary Settings (8, 14, 21) edit window

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select the boundary condition.

Load the Library coefficient using the path Heat Transfer Coefficients > Air, Ext. Natural Convection as shown in Table 5.34.

Enter L_wall in the Heat transfer coefficient (h) expression in place of the L_htgh term for boundary 28:

h_ave(T[1/K],Tinf_htgh[1/K],L_wall[1/m])[W/(m^2*K)]

Table 5.34 Boundary Settings–General Heat Transfer Edit Window

Boundary	Boundary Condition	Library Coefficient	Figure Number
28	Heat flux	Nat. Vertical wall, L=height	5.95
34	Heat flux	Nat. Horiz. plane, Upside, L=width	5.96

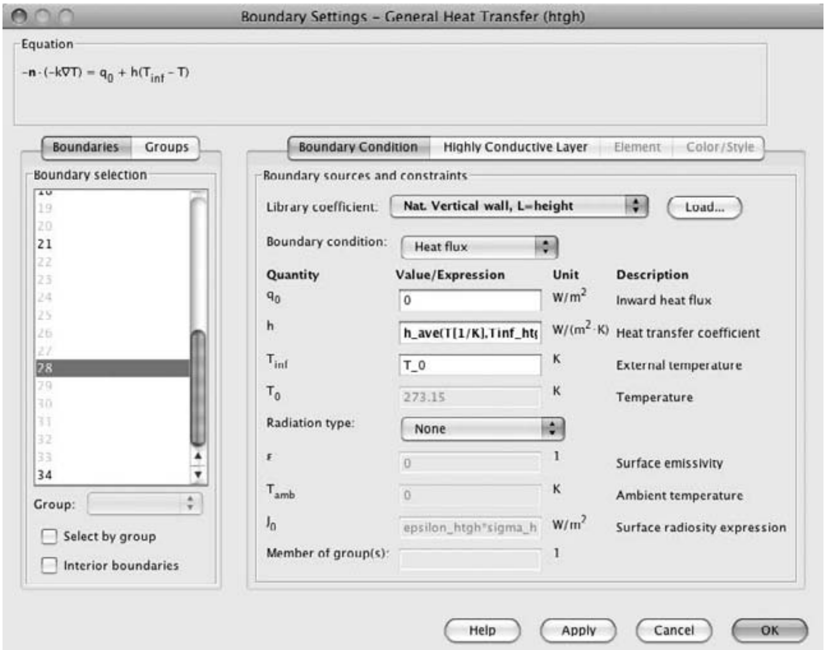


FIGURE 5.95 2D axisymmetric Thermos_Container_2 model Boundary Settings (28) edit window

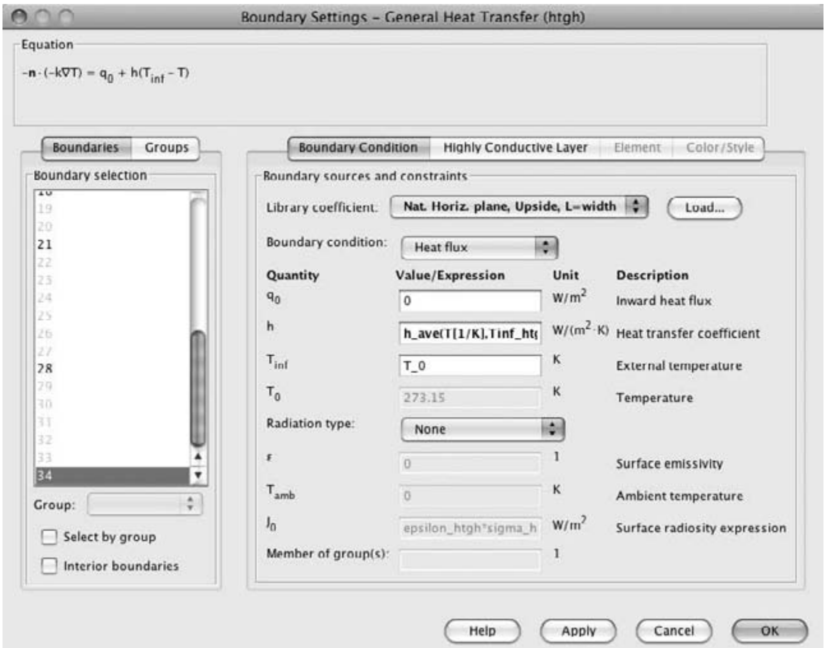


FIGURE 5.96 2D axisymmetric Thermos_Container_2 model Boundary Settings (34) edit window

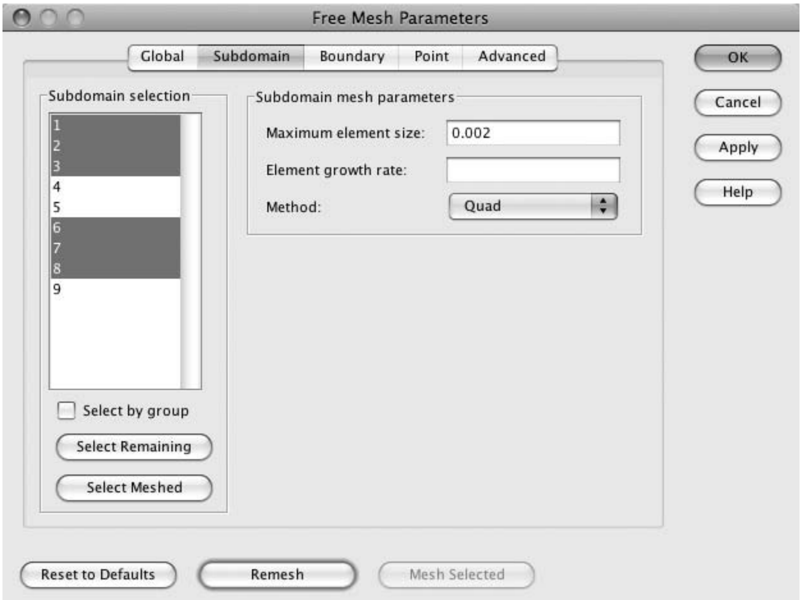


FIGURE 5.97 2D axisymmetric Thermos_Container_2 model Free Mesh Parameters edit window

Enter T_0 in the External temperature (T_{inf}) edit window for boundary 28. See Figure 5.95.

Enter L_top in the Heat transfer coefficient (h) expression in place of the L_htgh term for boundary 34:

h_ave(T[1/K],Tinf_htgh[1/K],L_top[1/m])[W/(m^2*K)]

Enter T_0 in the External temperature (T_{inf}) edit window for boundary 34. See Figure 5.96.

Click OK.

Mesh Generation

From the menu bar, select Mesh > Free Mesh Parameters > Subdomain 1, 2, 3, 6, 7, 8. Enter Maximum element size 0.002. Select Method > Quad. Click the Remesh button. See Figure 5.97.

Click OK. See Figure 5.98.

Solving the 2D Axisymmetric Thermos_Container_2 Model

From the menu bar, select Solve > Solver Parameters > Parametric. Enter T_L in the Parameter name edit window. Enter 273.15:9.5:368.15 in the Parameter values edit window. See Figure 5.99. Click OK.

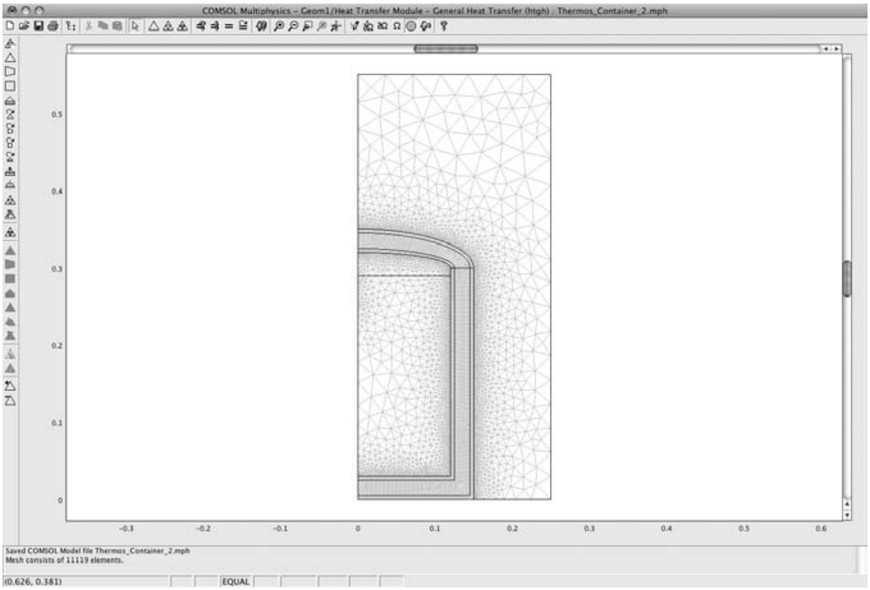


FIGURE 5.98 2D axisymmetric Thermos_Container_2 model mesh

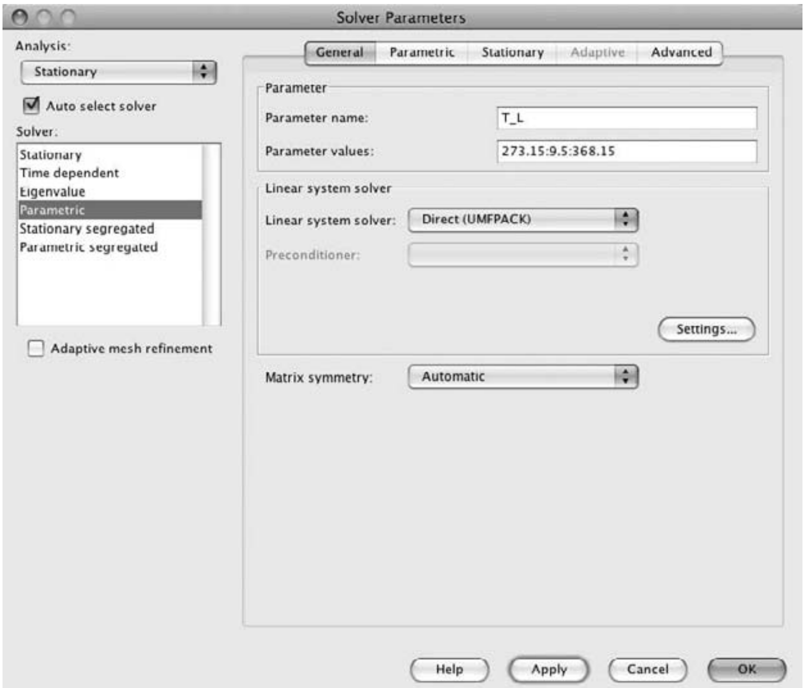


FIGURE 5.99 2D axisymmetric Thermos_Container_2 model Solver Parameters edit window

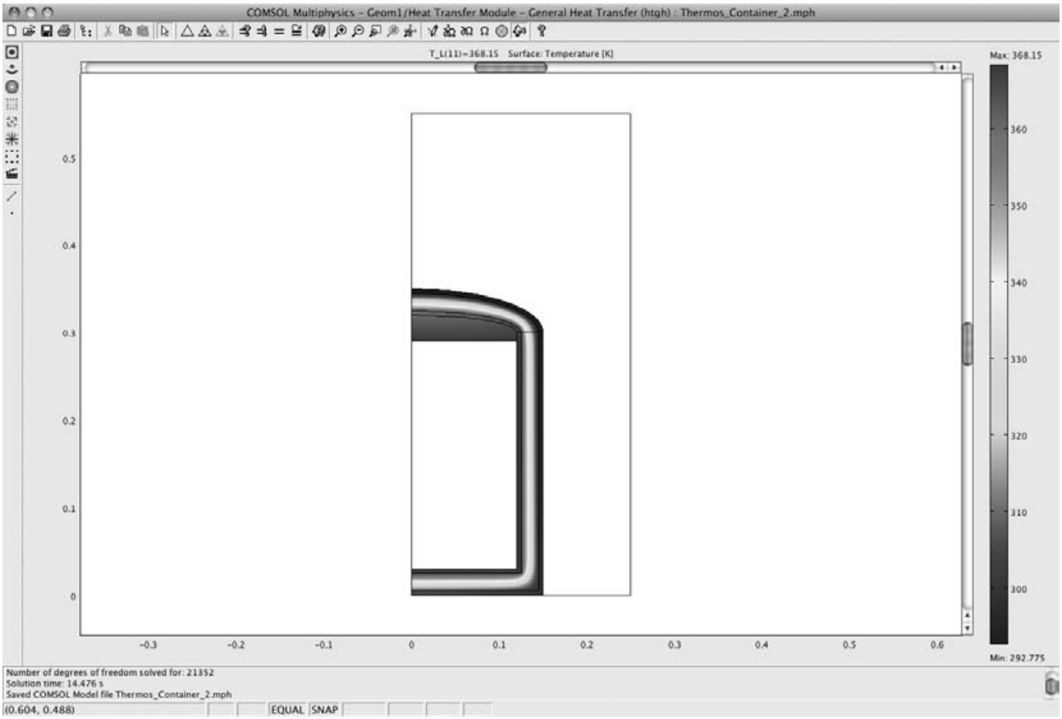


FIGURE 5.100 2D axisymmetric Thermos_Container_2 model using the Parametric Solver (UMFPACK)

From the menu bar, select Solve > Solve Problem. See Figure 5.100.

Postprocessing

Select Postprocessing > Plot Parameters > Surface. Select "°C [degC]" in the Unit pull-down list. See Figure 5.101.

Click OK. See Figure 5.102.

Given that our main interest in creating the 2D axisymmetric Thermos_Container_2 model is to examine the heat transfer, the next step is to determine the heat loss. Select Postprocessing > Boundary Integration. Select boundaries 28 and 34 (the wall and top of the tank, respectively). Select "Normal total heat flux" in the Predefined quantities pull-down list. Check the Compute surface integral (for axisymmetric modes) check box. See Figure 5.103.

Click OK. The result of the Boundary Integration (~37 W) is displayed as Value of surface integral: xx.xxxx [W], Expression: ntflux_htgh, Boundaries: 28, 34 in the display window at the bottom of the COMSOL user interface. The amount of energy lost is approximately 45% of that lost using the urethane foam insulation (~82 W) in the 2D axisymmetric Thermos_Container_1 model. See Figure 5.104.

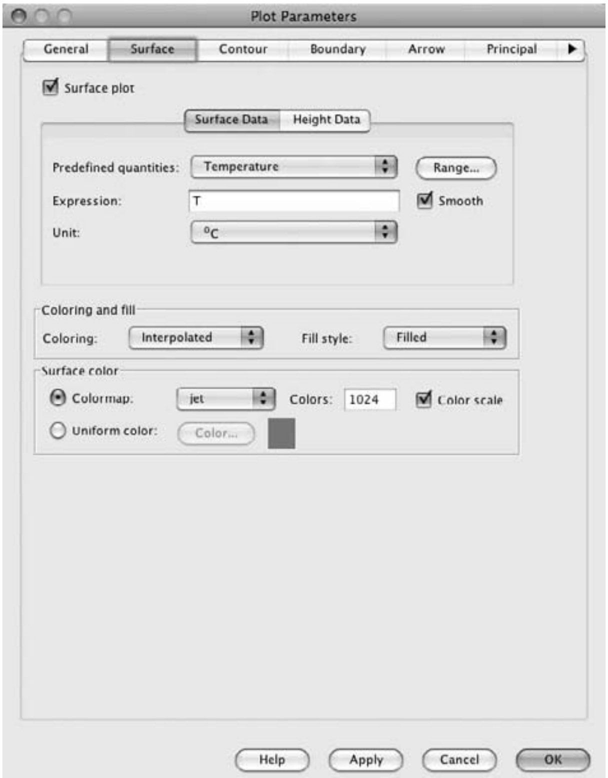


FIGURE 5.101 2D axisymmetric Thermos_Container_2 model Plot Parameters window

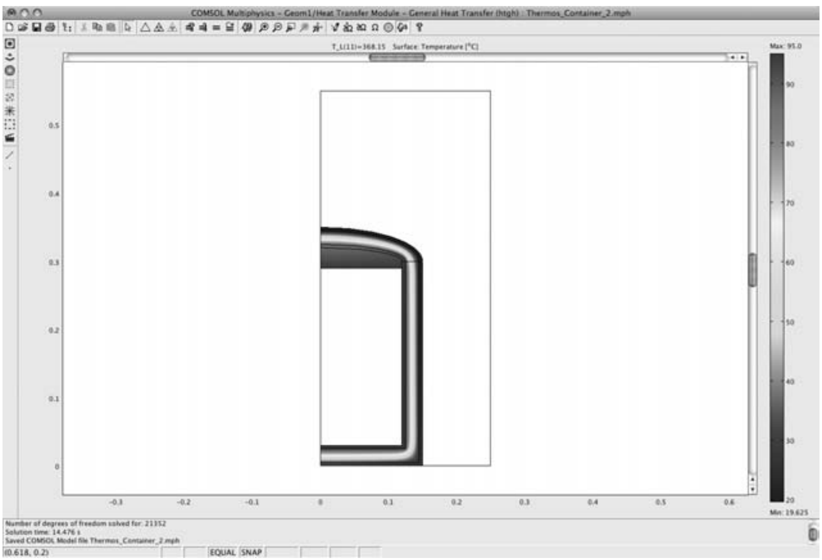


FIGURE 5.102 2D axisymmetric Thermos_Container_2 model surface temperature (°C)

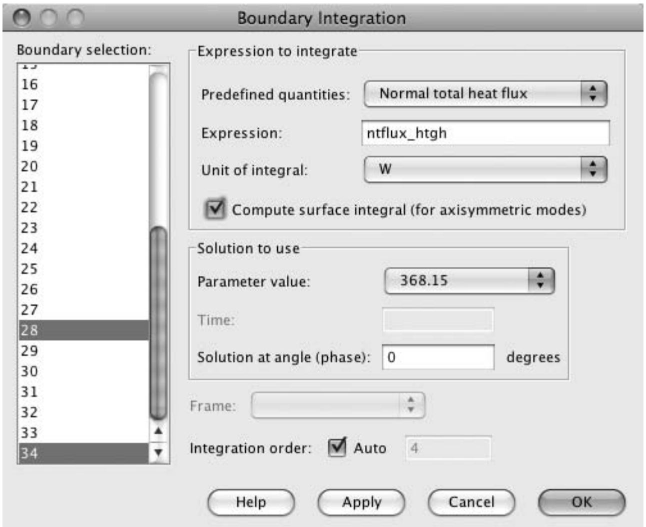


FIGURE 5.103 2D axisymmetric Thermos_Container_2 model Boundary Integration edit window

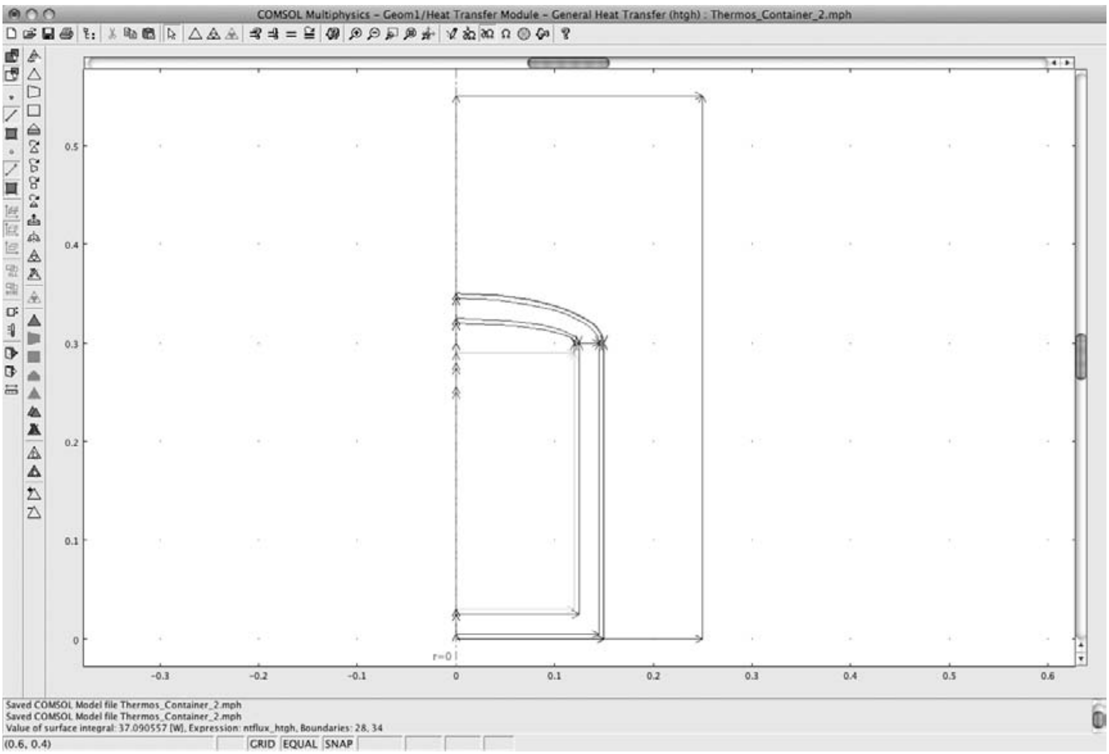


FIGURE 5.104 2D axisymmetric Thermos_Container_2 model user interface display window

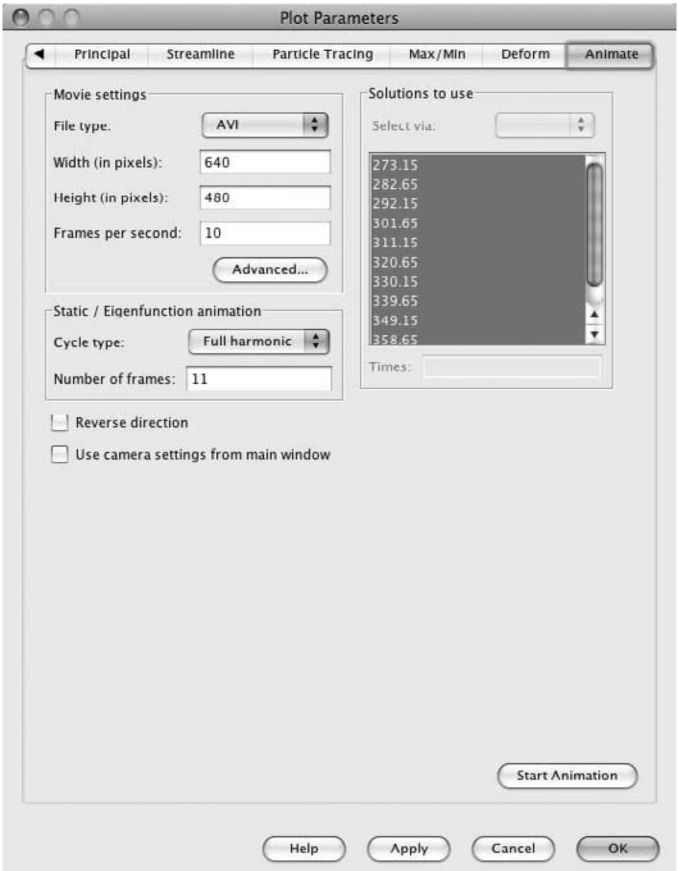


FIGURE 5.105 2D axisymmetric Thermos_Container_2 model Plot Parameters window

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.105.

Click the Start Animation button. See Figure 5.106.

Alternatively, you can play the file Movie5_TC_2.avi that was supplied with this book.

Second Variation on the 2D Axisymmetric Thermos_Container Model

NOTE In this model, the second variation on the 2D axisymmetric Thermos_ Container model, a glass material replaces the stainless steel walls and a vacuum cavity replaces the urethane foam.

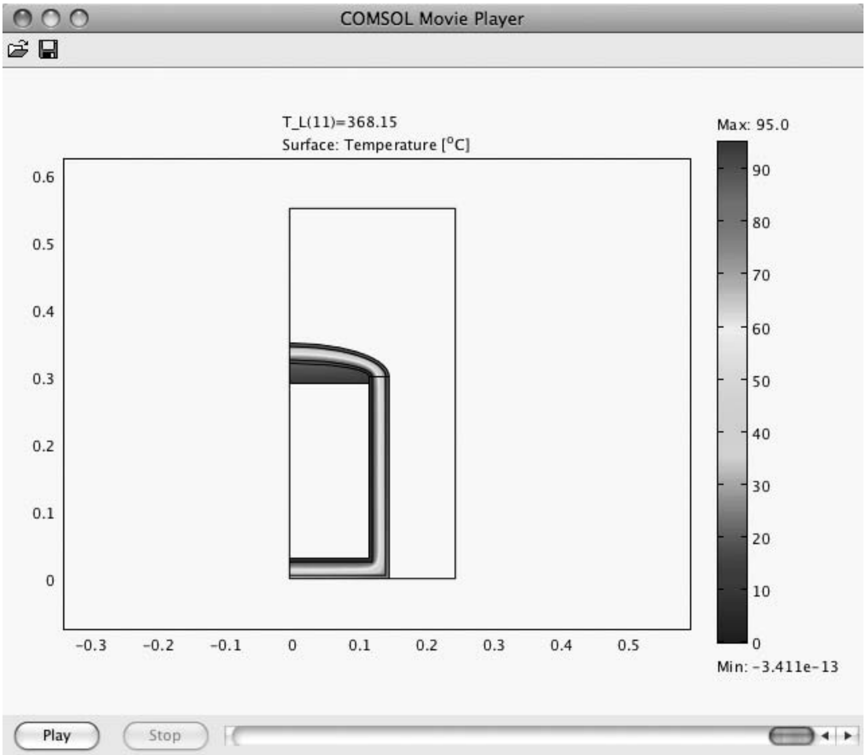


FIGURE 5.106 2D axisymmetric Thermos_Container_2 model animation, final frame

To start building the Thermos_Container_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > General Heat Transfer > Steady-state analysis. See Figure 5.107. Click OK.

Constants

NOTE When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.35; also see Figure 5.108. Click OK.

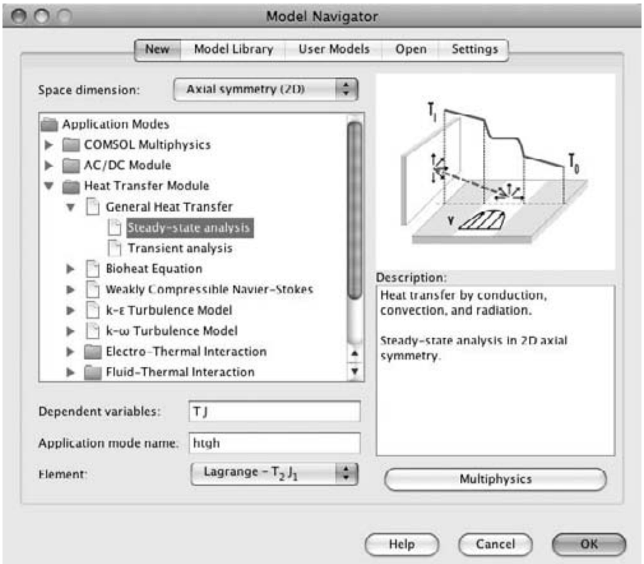


FIGURE 5.107 2D axisymmetric Thermos_Container_3 Model Navigator setup

Table 5.35 Constants Edit Window

Name	Expression	Description
p_0	1.0[atm]	Air pressure
p_vac	1.33e-7[Pa]	Pressure in vacuum
T_0	2.7315e2[K]	Boundary temperature
T_L	9.5e1[degC]	Liquid temperature
L_wall	0.35[m]	Projected height of tank wall
L_top	0.15[m]	Width of top

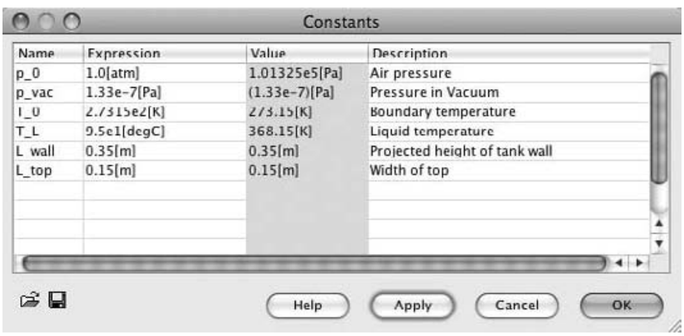


FIGURE 5.108 2D axisymmetric Thermos_Container_3 model Constants edit window

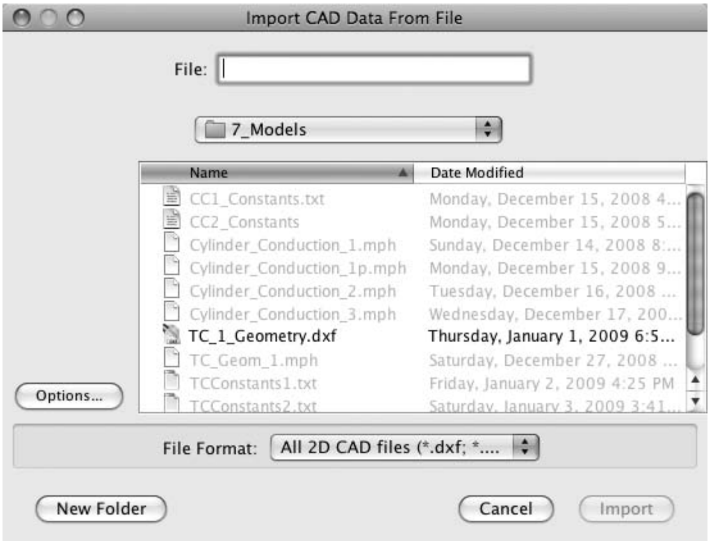


FIGURE 5.109 2D axisymmetric Thermos_Container_2 model import

---

NOTE When building a model, it is usually best to choose a name for modeler-defined parameters (e.g., constants, scalar functions) that are easily recalled and associated with the function that they provide in the model (e.g., p_vac, L_wall).

Select File > Save as. Enter Thermos_Container_3. Click the Save button.

Importing the 2D Axisymmetric Thermos Container

NOTE The actual sequence of steps required in the building of the 2D axisymmetric Thermos_Container was presented in the discussion of the 2D axisymmetric Thermos_Container_1 model. Now the modeler can use the import function to utilize the same physical model configuration and explore the influence of different materials and materials properties on the overall model design behavior.

Using the menu bar, select File > Import > CAD Data From File. Select "TC_1_Geometry.dxf." See Figure 5.109. Click the Import button.

NOTE Because the Geometry.dxf file contains only boundary information, the modeler needs to create a domain to which the boundary information can be applied.

Using the menu bar, select Draw > Specify Objects > Rectangle, as indicated in Table 5.36.

Click OK. See Figure 5.110.

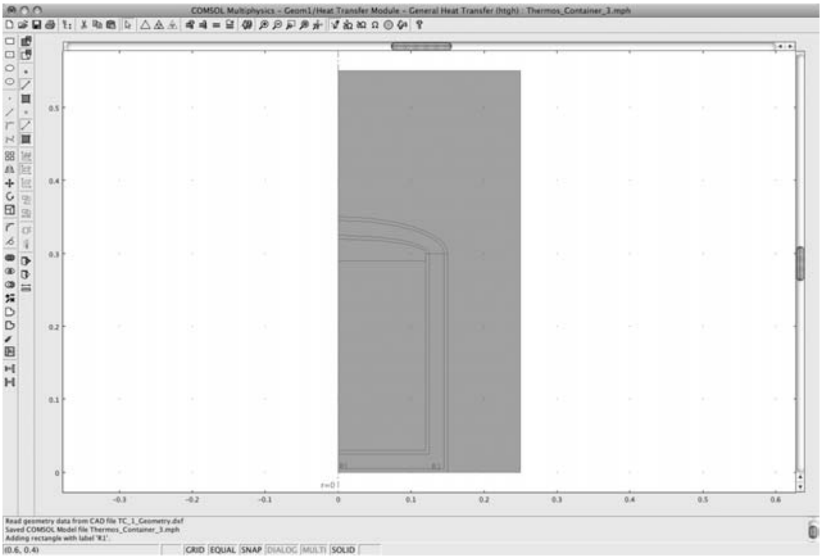


FIGURE 5.110 2D axisymmetric Thermos_Container_3 model import and rectangle R1

Physics Subdomain Settings: General Heat Transfer

Having established the geometry for the 2D axisymmetric Thermos_Container_3 model, the next step is to define the fundamental Physics conditions. Select Physics > Subdomain Settings. In the Subdomain edit windows, enter the information shown in Table 5.37. See Figure 5.111.

In the Subdomain edit windows, enter the information shown in Table 5.38.

Table 5.36 Rectangle Edit Window

Object Number	Width	Height	Base	*r*	*z*
Rectangle 1	0.25	0.55	Corner	0	0

Table 5.37 Subdomain Edit Window

Subdomain	Operation	Name
1, 3, 6, 8	Load	Basic Materials Properties > Silica Glass

Table 5.38 Subdomain Edit Window

Subdomain	Operation	Name
2, 7	Load	Basic Materials Properties > Air, 1 atm

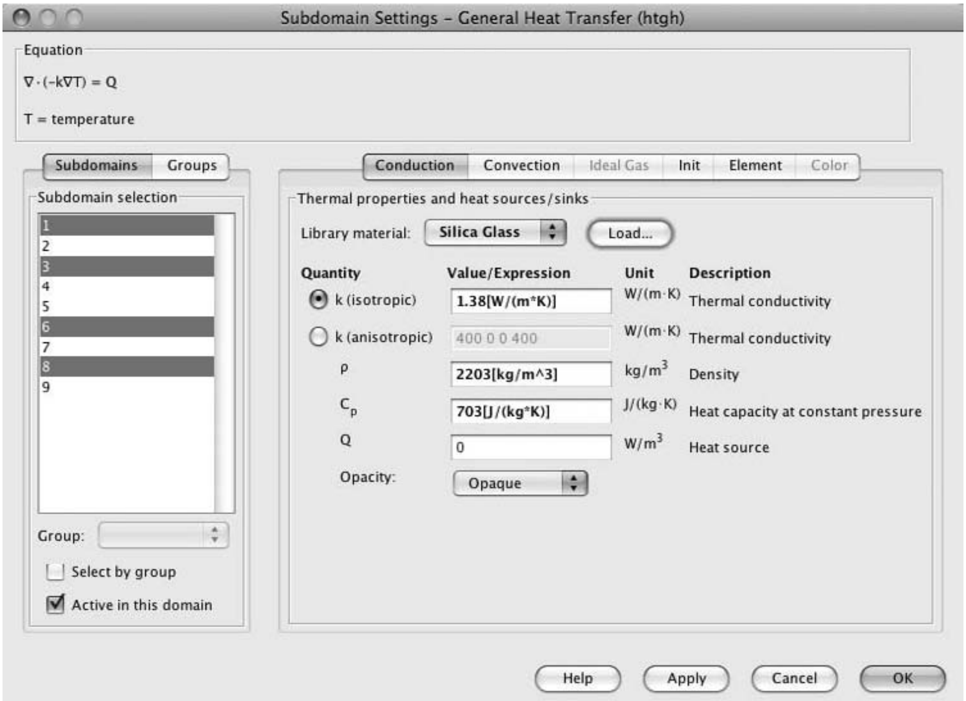


FIGURE 5.111 2D axisymmetric Thermos_Container_3 model Subdomain Settings (1, 3, 6, 8) edit window

Enter p_vac in place of p in the density expression rho(p[1/Pa],T[1/K])[kg/m^3], so that it reads rho(p_vac[1/Pa],T[1/K])[kg/m^3] in the Density edit window. Select "Transparent" from the Opacity pull-down list. See Figure 5.112.

In the Subdomain edit windows, enter the information shown in Table 5.39.

Enter p_0 in place of p in the density expression to yield rho(p_0[1/Pa], T[1/K])[kg/m^3] in the Density edit window. Select "Transparent" from the Opacity pull-down list. See Figure 5.113.

In the Subdomain edit windows, enter the information shown in Table 5.40. See Figure 5.114.

Table 5.39 Subdomain Edit Window

Subdomain	Operation	Name
5, 9	Load	Basic Materials Properties > Air, 1 atm

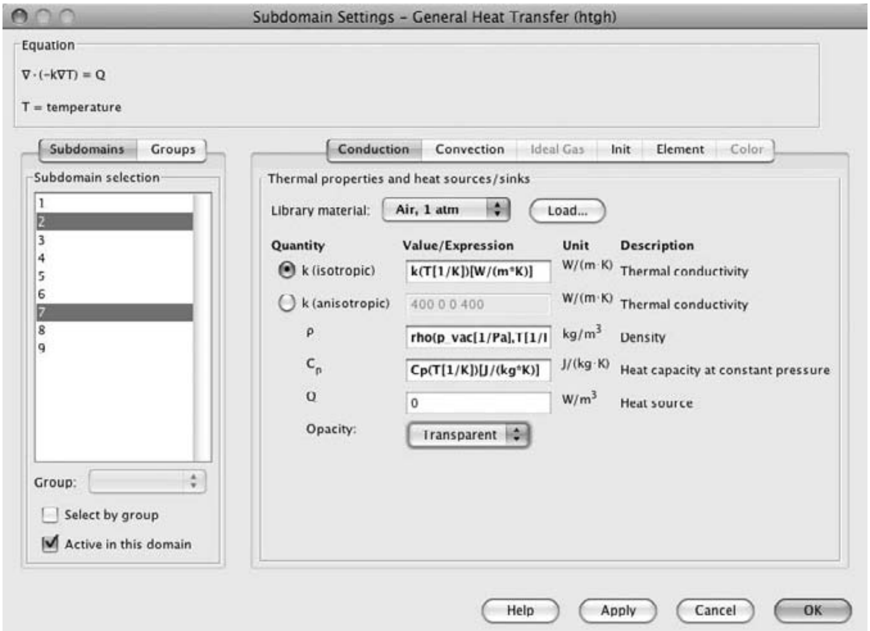


FIGURE 5.112 2D axisymmetric Thermos_Container_3 model Subdomain Settings (2, 7) edit window

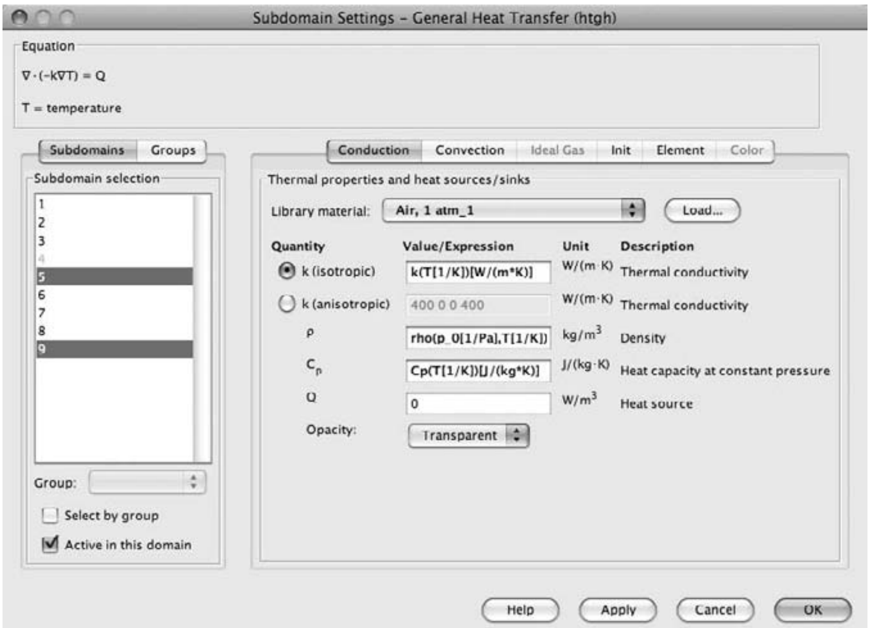


FIGURE 5.113 2D axisymmetric Thermos_Container_2 model Subdomain Settings (5, 9) edit window

Table 5.40 Subdomain Edit Window

Subdomain	Operation	Name
4	Load	Basic Materials Properties > Water, liquid

NOTE In this model, because of the high thermal conductivity of water, the temperature of the liquid is very close to uniform throughout. Thus uniformity can be assumed and subdomain 4 can be made inactive. The temperature of the liquid is incorporated into the boundary conditions. Also, because this model will calculate only the heat transfer, and not the detailed convection flow in the surrounding air, subdomain 9 can be made inactive. Convection losses are incorporated into the heat transfer coefficient boundary conditions. Incorporating both of these assumptions into the model significantly simplifies the model calculations.

Select Subdomain 4. Uncheck the Active in this domain check box. Select Subdomain 9. Uncheck the Active in this domain check box. Click OK.

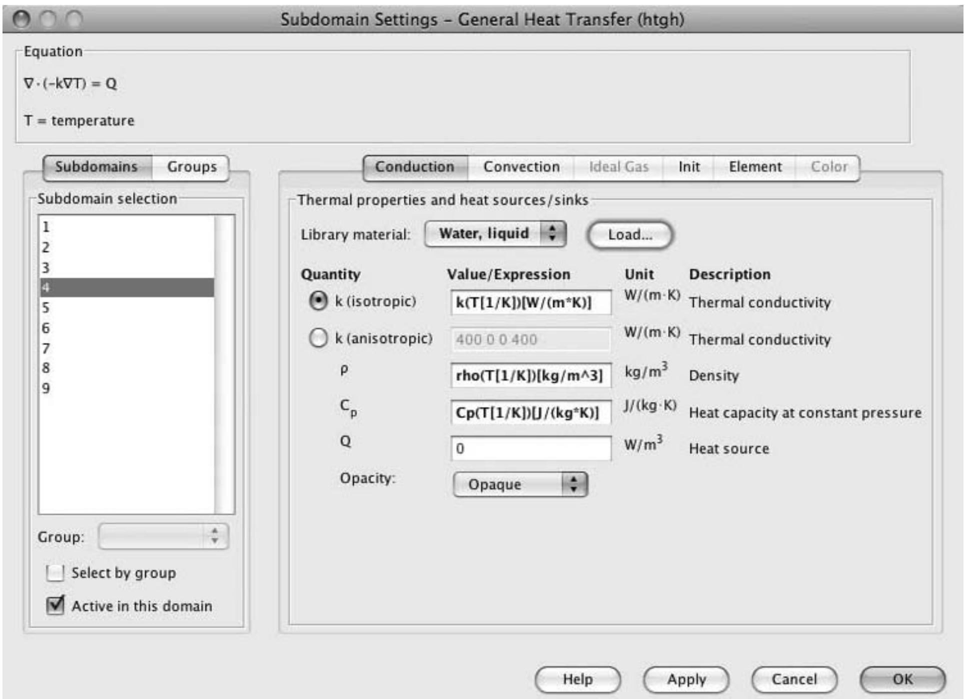


FIGURE 5.114 2D axisymmetric Thermos_Container_3 model Subdomain Settings (4) edit window

Table 5.41 Boundary Settings–General Heat Transfer Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
2	Thermal insulation	—	5.115
8, 14, 21	Temperature	T_L	5.116

Physics Boundary Settings: General Heat Transfer

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 5.41. Click OK. See Figures 5.115 and 5.116.

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select the Boundary condition.

Load the Library coefficient using the path Heat Transfer Coefficients > Air, Ext. Natural Convection as shown in Table 5.42.

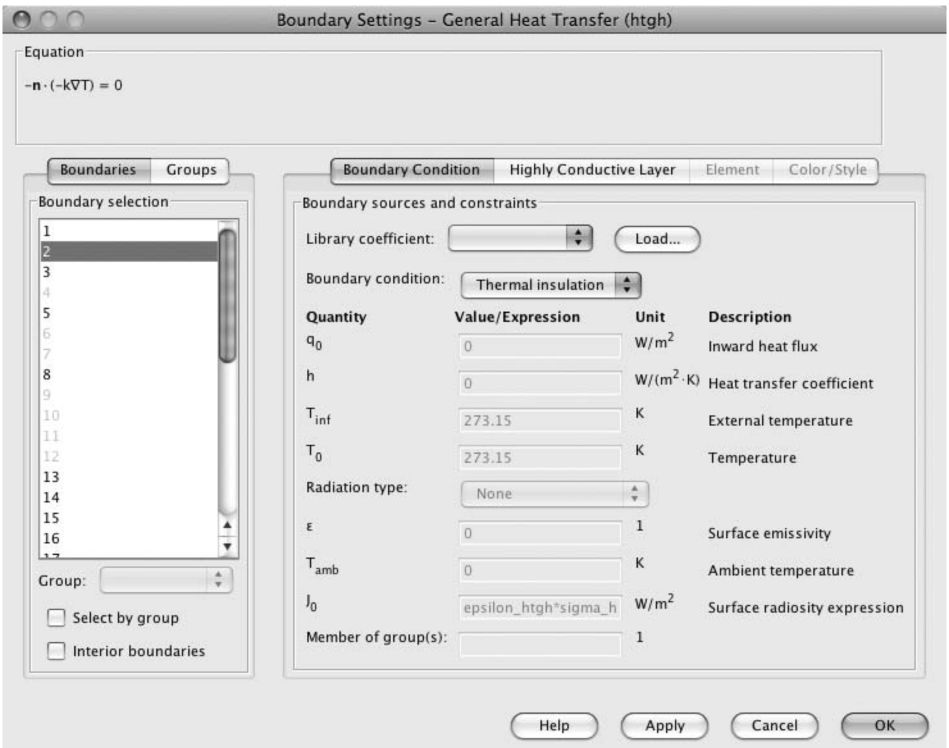


FIGURE 5.115 2D axisymmetric Thermos_Container_3 model Boundary Settings (2) edit window

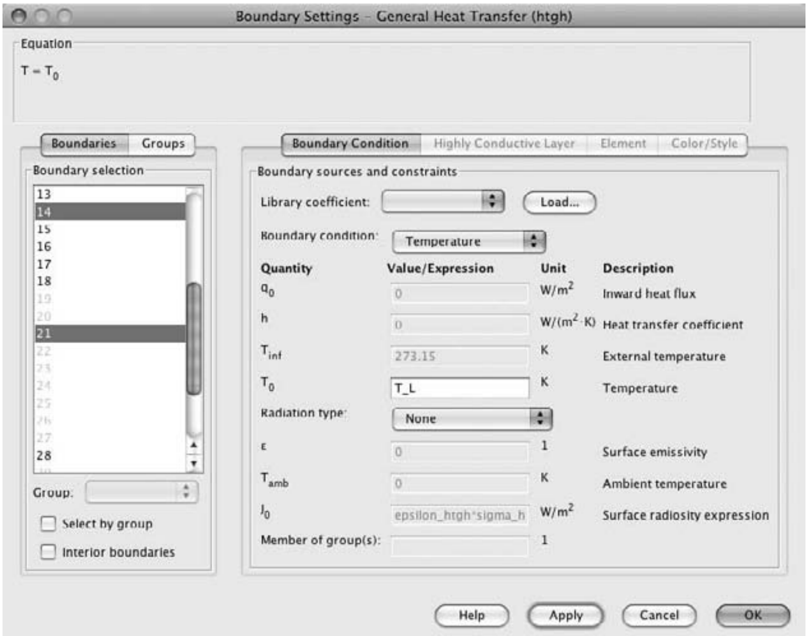


FIGURE 5.116 2D axisymmetric Thermos_Container_3 model Boundary Settings (8, 14, 21) edit window

Enter L_wall in the Heat transfer coefficient (h) expression in place of the L_htgh term for boundary 28:

h_ave(T[1/K],Tinf_htgh[1/K],L_wall[1/m])[W/(m^2*K)]

Enter T_0 in the External temperature (T_{inf}) edit window for boundary 28. See Figure 5.117.

Enter L_top in the Heat transfer coefficient (h) expression in place of the L_htgh term for boundary 34:

h_ave(T[1/K],Tinf_htgh[1/K],L_top[1/m])[W/(m^2*K)]

Enter T_0 in the External temperature (T_{inf}) edit window for boundary 34. See Figure 5.118.

Click OK.

Table 5.42 Boundary Settings–General Heat Transfer Edit Window

Boundary	Boundary Condition	Library Coefficient	Figure Number
28	Heat flux	Nat. Vertical wall, L=height	5.117
34	Heat flux	Nat. Horiz. plane, Upside, L=width	5.118

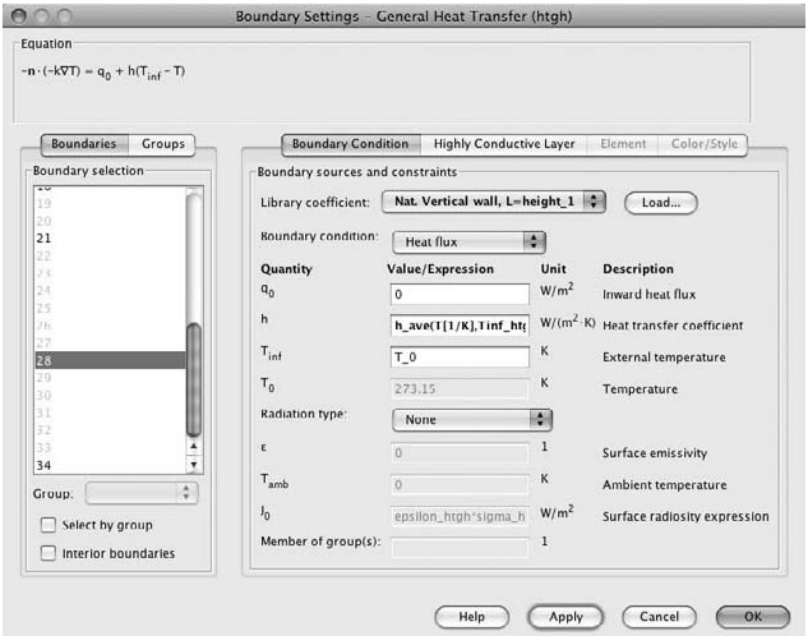


FIGURE 5.117 2D axisymmetric Thermos_Container_3 model Boundary Settings (28) edit window

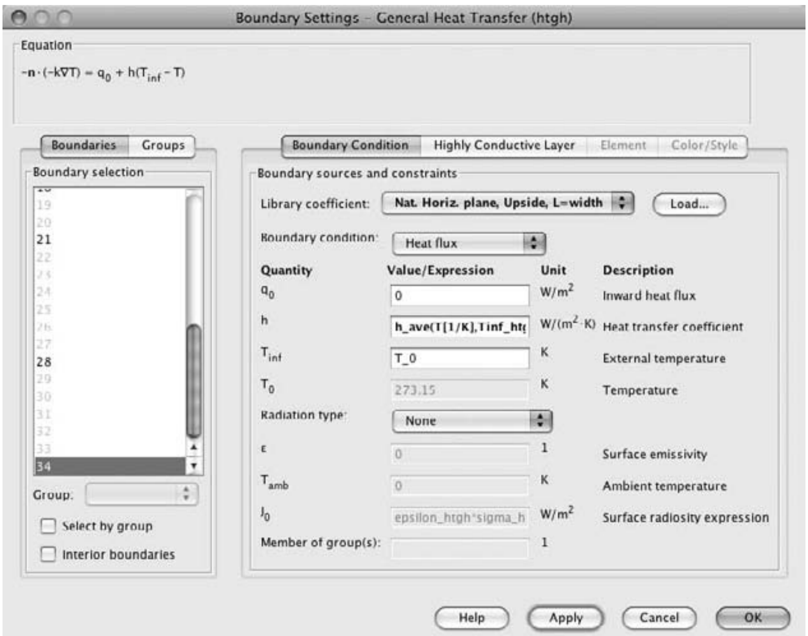


FIGURE 5.118 2D axisymmetric Thermos_Container_3 model Boundary Settings (34) edit window

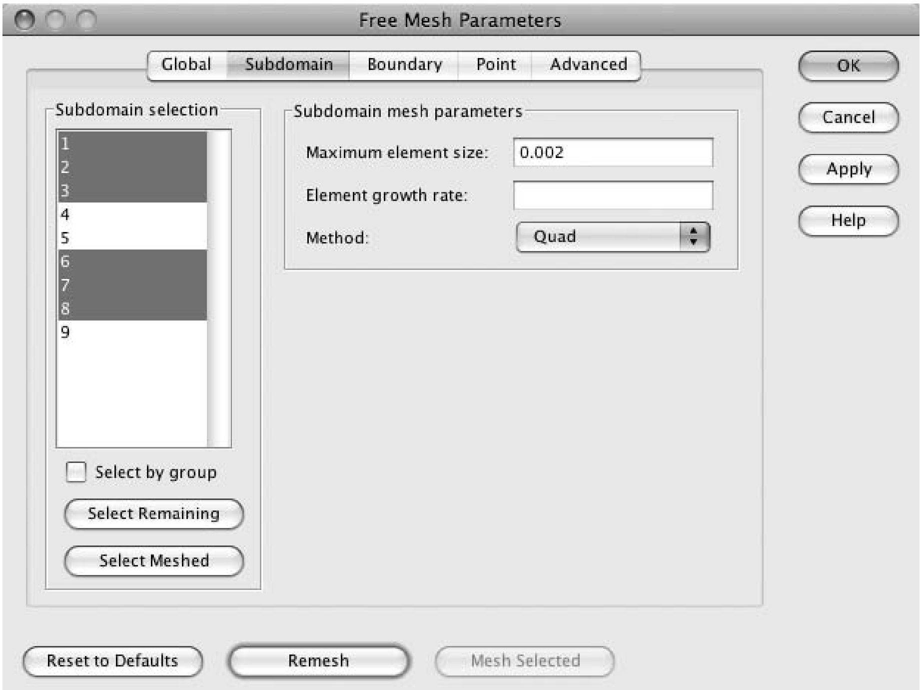


FIGURE 5.119 2D axisymmetric Thermos_Container_3 model Free Mesh Parameters edit window

Mesh Generation

From the menu bar, select Mesh > Free Mesh Parameters > Subdomain 1, 2, 3, 6, 7, 8. Enter Maximum element size 0.002. Select Method > Quad. Click the Remesh button. See Figure 5.119.

Click OK. See Figure 5.120.

Solving the 2D Axisymmetric Thermos_Container_3 Model

From the menu bar, select Solve > Solver Parameters > Parametric. Enter T_L in the Parameter name edit window. Enter 273.15:9.5:368.15 in the Parameter values edit window. See Figure 5.121. Click OK.

From the menu bar, select Solve > Solve Problem. See Figure 5.122.

Postprocessing

Select Postprocessing > Plot Parameters > Surface. Select "°C [degC]" in the Unit pull-down list. See Figure 5.123.

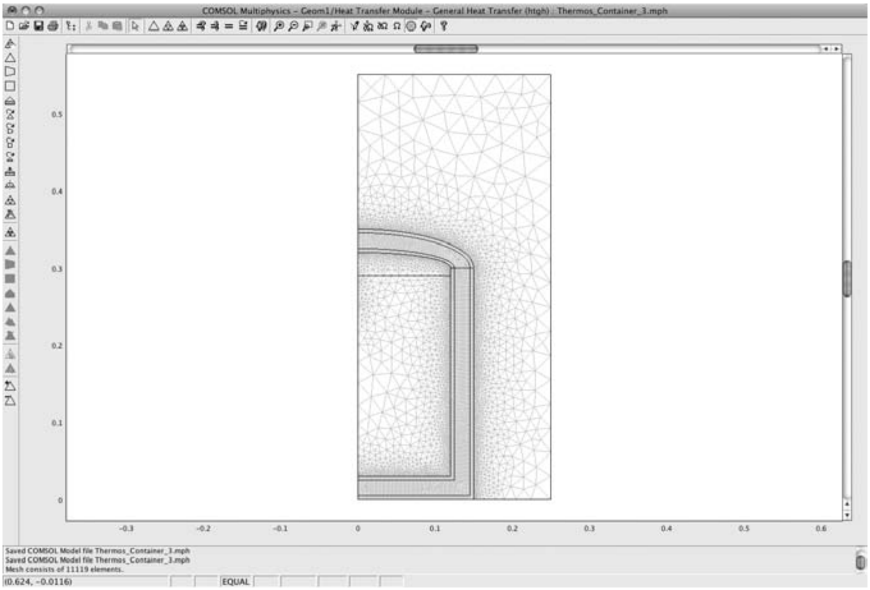


FIGURE 5.120 2D axisymmetric Thermos_Container_3 model mesh

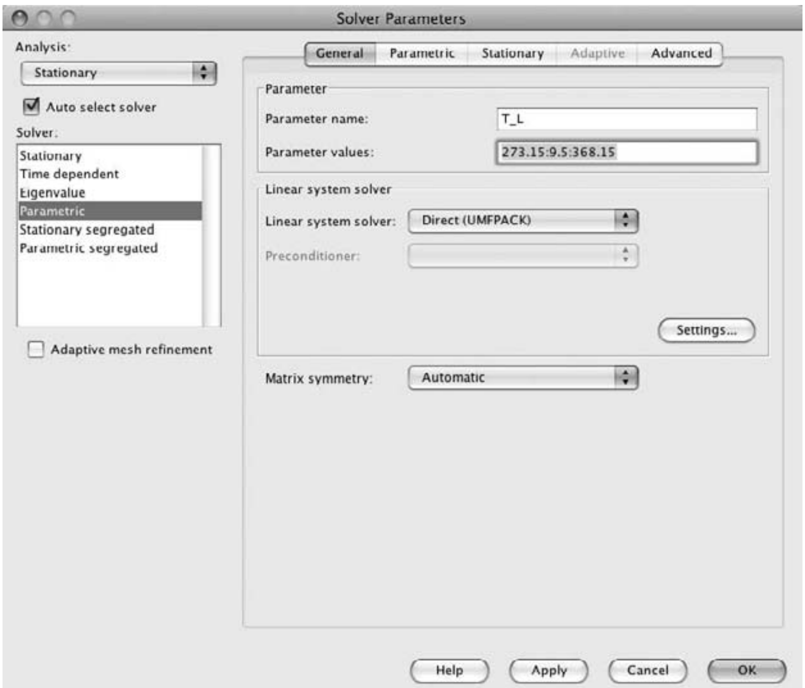


FIGURE 5.121 2D axisymmetric Thermos_Container_3 model Solver Parameters edit window

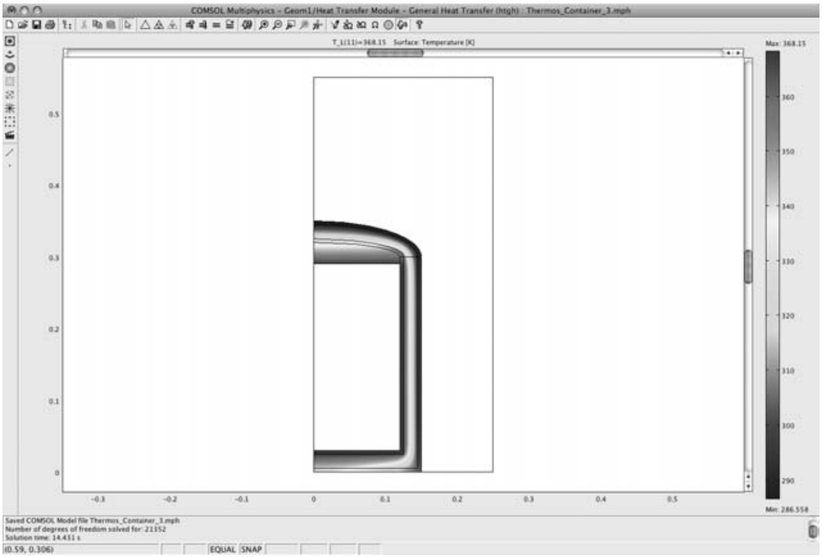


FIGURE 5.122 2D axisymmetric Thermos_Container_3 model using the Parametric Solver (UMFPACK)

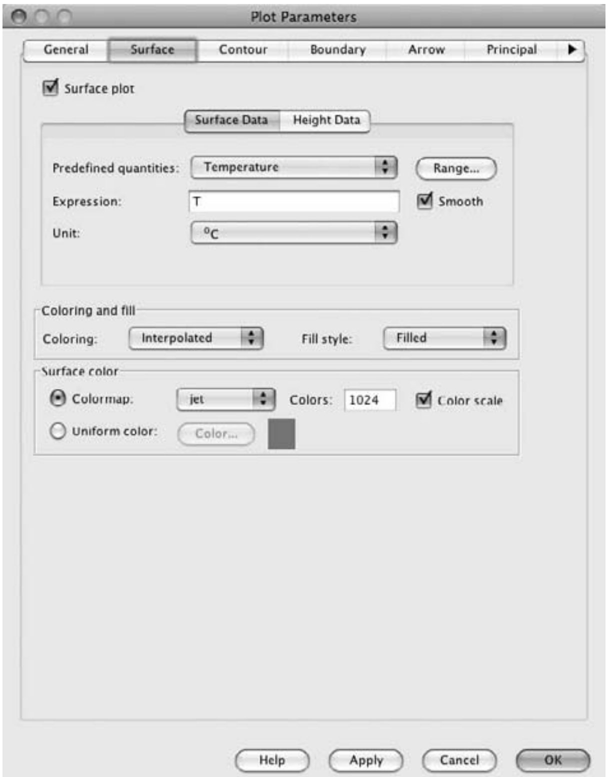


FIGURE 5.123 2D axisymmetric Thermos_Container_3 model Plot Parameters window

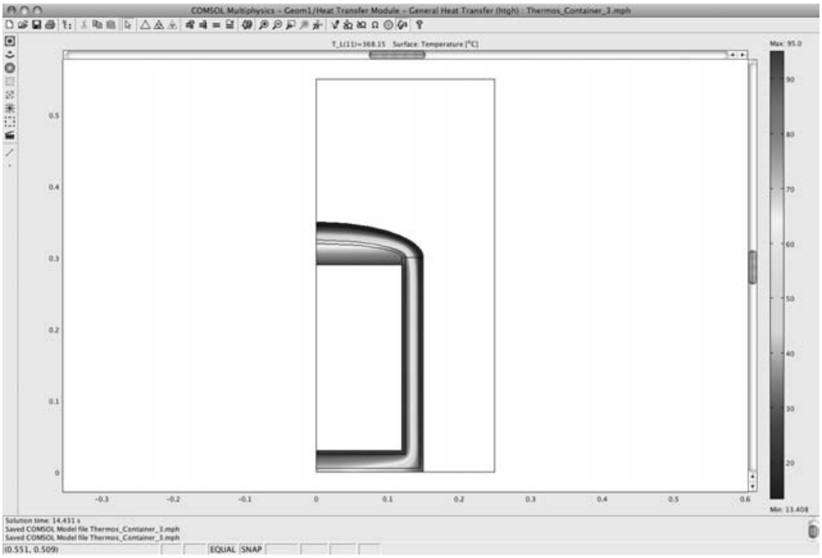


FIGURE 5.124 2D axisymmetric Thermos_Container_3 model surface temperature (°C)

Click OK. See Figure 5.124.

Given that our main interest in creating the 2D axisymmetric Thermos_Container_3 model is to examine the heat transfer, the next step is to determine the heat loss. Select Postprocessing > Boundary Integration. Select boundaries 28 and 34 (the wall and top of the tank, respectively). Select "Normal total heat flux" in the Predefined quantities pull-down list. Check the Compute surface integral (for axisymmetric modes) check box. See Figure 5.125.

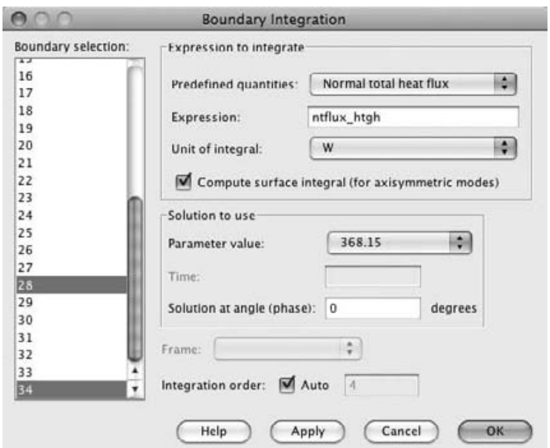


FIGURE 5.125 2D axisymmetric Thermos_Container_3 model Boundary Integration edit window

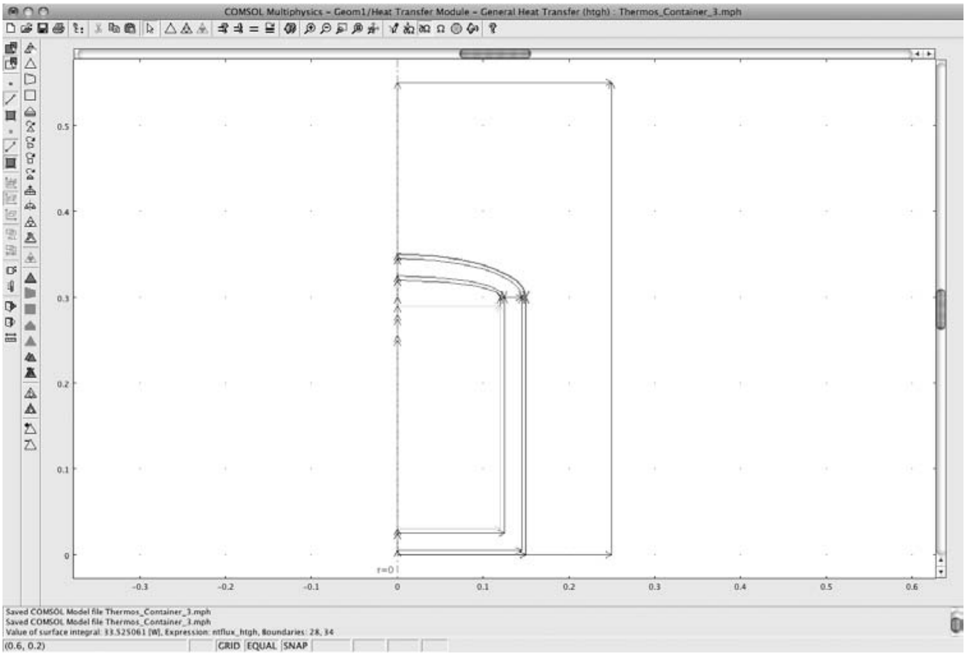


FIGURE 5.126 2D axisymmetric Thermos_Container_3 model user interface display window

Click OK. The result of the Boundary Integration (~34 W) is displayed as Value of surface integral: xx.xxxx [W], Expression: ntflux_htgh, Boundaries: 28, 34 in the display window at the bottom of the COMSOL user interface. The amount of energy lost is approximately 41% of that lost using the urethane foam insulation in the 2D axisymmetric Thermos_Container_1 model (~82 W). See Figure 5.126.

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.127.

Click the Start Animation button. See Figure 5.128.

Alternatively, you can play the file Movie5_TC_3.avi that was supplied with this book.

2D Axisymmetric Thermos_Container Models: Summary and Conclusions

The models presented in this section of Chapter 5 have introduced the following concepts: two-dimensional axisymmetric modeling (Axial symmetry [2D]), cylindrical coordinates, conductive media DC, Heat Transfer Module, heat conduction theory, opaque and transparent thermally conductive materials, export and import of CAD

FIGURE 5.127 2D axisymmetric Thermos_Container_3 model Plot Parameters window

drawings (.dxf files), heat transfer coefficients, and vacuum. Previously introduced concepts employed in this section include the triangular mesh, free mesh parameters, subdomain mesh, maximum element size, and quadrilateral mesh (quad).

A comparison of the calculated results for the three thermos container models is shown in Table 5.43. As can be readily observed, the presence of a vacuum cavity significantly reduces the rate of heat flow through the model and the associated heat loss.

Table 5.43 Thermos Container Modeling Results Summary

Model Number	Materials Used	Vacuum	Heat Loss	ΔW (%)
1	304ss, urethane foam	No	~82 W	—
2	304ss	Yes	~37 W	~45%
3	Silica glass	Yes	~34 W	~41%

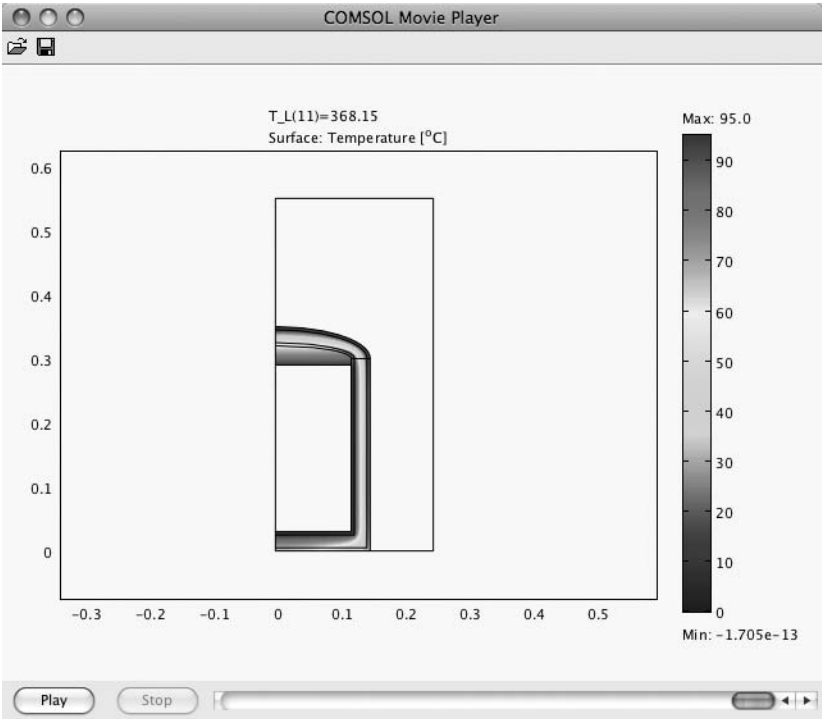


FIGURE 5.128 2D axisymmetric Thermos_Container_2 model animation, final frame

References

1. http://en.wikipedia.org/wiki/History_of_thermodynamics
2. http://en.wikipedia.org/wiki/William_Thomson,_1st_Baron_Kelvin
3. http://en.wikipedia.org/wiki/James_Prescott_Joule
4. http://en.wikipedia.org/wiki/Ludwig_Boltzmann
5. http://en.wikipedia.org/wiki/James_Clerk_Maxwell
6. http://en.wikipedia.org/wiki/Max_Planck
7. http://en.wikipedia.org/wiki/Issac_newton
8. http://en.wikipedia.org/wiki/Newton%27s_Law_of_Cooling#Newton.27s_law_of_cooling
9. http://en.wikipedia.org/wiki/Joseph_Fourier
10. http://en.wikipedia.org/wiki/Fourier%27s_Law
11. A. D. Cameron, J. A. Casey, and G. B. Simpson, NAFEMS Benchmark Tests for Thermal Analysis (Summary), NAFEMS Ltd., 1986.

12. J. R. Davis, ed., *Metals Handbook Desk Edition,* second edition, ASM International, 1998.
13. http://en.wikipedia.org/wiki/Niobium
14. http://en.wikipedia.org/wiki/Sir_James_Dewar
15. http://en.wikipedia.org/wiki/Royal_Society
16. http://en.wikipedia.org/wiki/Thermos
17. ASM International, *Engineered Materials Desk Edition,* "Thermal Analysis and Properties of Polymers," Table 22, Thermal and related properties of amino resins (urethane foam).
18. http://www.matweb.com/ (glass)
19. http://www.matweb.com/ (304 stainless steel)

Exercises

1. Build, mesh, and solve the COMSOL 2D axisymmetric cylinder conduction model problem presented in this chapter.
2. Build, mesh, and solve the first variation of the 2D axisymmetric cylinder conduction model problem presented in this chapter.
3. Build, mesh, and solve the second variation of the 2D axisymmetric cylinder conduction model problem presented in this chapter.
4. Build, mesh, and solve the 2D axisymmetric Thermos_Container model presented in this chapter.
5. Build, mesh, and solve the first variation of the 2D axisymmetric Thermos_Container model presented in this chapter.
6. Build, mesh, and solve the second variation of the 2D axisymmetric Thermos_Container model presented in this chapter.
7. Explore other variations of the arguments in the COMSOL 2D axisymmetric cylinder conduction models.
8. Explore other variations of the arguments in the 2D axisymmetric Thermos_Container models.
9. Explore how an increase in the pressure modifies the behavior of the COMSOL 2D axisymmetric cylinder conduction model.
10. Explore how changes in the tank geometry affect the heat loss in the 2D axisymmetric Thermos_Container model.

6 2D Simple Mixed-Mode Modeling

In This Chapter

■ 2D Mixed-Mode Guidelines for New COMSOL® Multiphysics® Modelers

2D Mixed-Mode Modeling Considerations

It is assumed, at this point, that the reader has been exposed, at least briefly, to the information contained in Chapters 4 and 5. In this chapter, the basic material from Chapters 4 and 5 is utilized and somewhat expanded. In the earlier chapters, models were built and then solved using a quasi-static approach. In this chapter, the transient (time-dependent) method of solution is introduced. Transient models are intrinsically more difficult than quasi-static models. Transient models require a firmer understanding of the underlying physics and a more complete characterization of the materials employed in the model.

NOTE In transient or time-dependent (e.g., dynamic, unsteady) models, at least one of the dependent variables changes as a function of time.

These 2D models implicitly assume, in compliance with the laws of physics, that the energy flow, the materials properties, the environment, and any other conditions and variables that are of interest are homogeneous, isotropic, and constant unless otherwise specified (e.g., time dependent) throughout the entire domain of interest, both within the model and, through the boundary conditions, in the environs of the model.

In the two models presented in this chapter, the resistive heating model and the inductive heating model, heat is generated within the body of the modeled materials through the same mechanism, Ohm's law[1] (i.e., Joule heating[2]), by two fundamentally different, but similar methods. In the resistive heating models, heat is generated by the flow of direct current (DC)[3] through the body in the models, resulting in Joule heating. As the body heats, the temperature rises. Because the resistivity depends on the temperature, the resistivity (conductivity) changes and consequently the amount of heat generated within the body changes, and so on.

In the inductive heating model, eddy currents (alternating currents [AC][4]) are induced within the material of the modeled body. Heat is generated by the flow of the induced alternating current within the body, generating Joule heating. As the body heat increases, the temperature rises. Similarly, because the resistivity depends on the temperature, the resistivity (conductivity) changes and consequently the amount of heat generated within the body changes, the temperature rises, and so on.

NOTE As mentioned in previous chapters, it is always preferable for the modeler to be able to accurately anticipate the expected behavior (results) of the model and to understand how those results should be presented. Never assume that the default values that are initially present when the model is first created will suit the needs of a new model. Always verify that the values employed in the model are the correct ones needed for that model. Calculated solution values that significantly deviate from the expected values or from comparison values measured in experimentally derived realistic models are probably indicative of one or more modeling errors either in the original model design, in the earlier model analysis, or in the understanding of the underlying physics, or are simply due to human error.

2D Coordinate System

Two different 2D coordinate systems are employed in the models that are built in this chapter. In the first set of models (resistive heating), the basic 2D coordinate system plus time is employed. The second set of models (inductive heating) employs the 2D axisymmetric coordinate system plus time. Each of the coordinate systems was chosen

to facilitate the modeler building the least difficult model necessary to achieve a reasonably accurate demonstration of the principles involved and achieve a good first approximation result.

NOTE Because it is completely impossible to accommodate all variable factors into any scientific or engineering problem larger than the two-body problem,[5] each scientific or engineering calculation yields an approximate result. A good first approximation result is derived from a calculation that yields an answer that allows the modeler to determine the degree of feasibility of an adequate solution to the problem in question, within the limits of tolerable variance (error). All of the nonmodeling parameters need to be estimated either by the modeler, his or her power structure, or his or her accountant.

The purpose of the models presented here is to demonstrate the application of the chosen modeling techniques to applied physical prototypes, using measured materials properties for commercially available materials. These first approximation result models can be modified and used by the modeler to build other exploratory candidate models to determine the feasibility of similar devices as part of a more complex development or analysis project.

In a steady-state solution to a 2D model, parameters can vary only as a function of position in space (x) and space (y) coordinates. Such a 2D model represents the parametric condition of the model in a time-independent mode (quasi-static). In a transient solution model, parameters can vary both by position in space (x) and space (y) and in time (t); see Figure 6.1.

The transient solution model is essentially a sequential collection of (quasi-static) solutions, except that one or more of the dependent variables ($f(T, t)$) has changed with time. The space coordinates (x) and (y) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values ($x_{min} <= x <= x_{max}$) and ($y_{min} <= y <= y_{max}$). The time coordinate (t) represents the range of values ($t_{min} <= t <= t_{max}$) from the beginning of the observation period (t_{min}) to the end of the observation period (t_{max}).

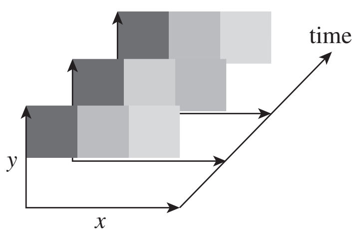


FIGURE 6.1 2D coordinate system, plus time

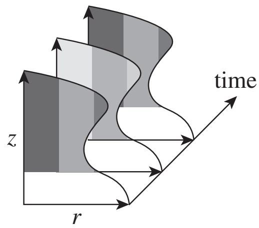


FIGURE 6.2 2D axisymmetric coordinate system, plus time

2D Axisymmetric Coordinate System

In the steady-state solution to any 2D axisymmetric model, parameters can vary only as a function of the radial position in space (r) and the axial position space (z) coordinates. Such a model represents the parametric condition of the model in a time-independent mode (quasi-static). In a transient solution model, parameters can vary both by position in space (r) and space (z) and in time (t); see Figure 6.2.

The transient solution model is essentially a sequential collection of steady-state (quasi-static) solutions. The space coordinates (r) and (z) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values ($r_{min} <= r <= r_{max}$) and ($z_{min} <= z <= z_{max}$). The time coordinate (t) represents the range of values ($t_{min} <= t <= t_{max}$) from the beginning of the observation period (t_{min}) to the end of the observation period (t_{max}).

Joule Heating and Heat Conduction Theory

Joule heating techniques are extremely important in device design considerations. Joule heating is applied to tasks as varied as heating houses (AC) and baking potatoes (microwave AC). It accounts for some of the most widely utilized technologies employed for research, design, and application in engineering and physics. Most modern products or processes require an understanding of Joule heating techniques either during development or during the use of the product or process (e.g., automobiles, plate glass fabrication, plastic extrusion, plastic products, houses, baked potatoes, ice cream).

NOTE Heating and heat transfer concerns have existed since the beginning of prehistory. There have been many contributors to our present understanding of the interaction of electric currents and solids. In this particular area, however, two scientists made especially notable contributions: Georg Ohm[6] and James Prescott Joule.[7] Ohm discovered Ohm's law:[8]

$$I = \frac{V}{R} \tag{6.1}$$

where I = current in amperes (A)
V = voltage (electromotive force) in volts (V)
R = resistance in ohms

Joule discovered Joule's law:[9]

$$Q = I^2 \cdot R \cdot t \tag{6.2}$$

where Q = heat generated in joules (J)
I = current in amperes (A)
R = resistance in ohms
t = time in seconds (S)

The first example presented in this chapter, the resistive heating model, explores the 2D electro-thermal interaction modeling of Joule heating using transient analysis. The model is solved for a material that is both electrically and thermally conductive. This model is implemented using the COMSOL® Multiphysics® Electro-Thermal Application Mode.

In the first variation on the resistive heating model, the new model is built to explore a common configurational change and is solved using the same COMSOL Multiphysics Application Mode. In the second variation on this model, a model is built that incorporates materials modifications in addition to the configurational changes; it is solved using the COMSOL Multiphysics AC/DC Electro-Thermal Application Mode. The second variation also explores the influence of a low-pressure gas/vacuum environment on the model's properties. The calculated modeling results are then compared.

The second example, the induced heating model, explores the use of induced AC eddy currents to create Joule heating in a 2D axisymmetric model. The first and second variations on the induced heating model explore the effects of materials and parametric changes.

Heat Conduction Theory

Heat conduction is a naturally occurring process that is readily observed in many aspects of modern life (e.g., refrigerators, freezers, microwave ovens, thermal ovens, engines). The heat transfer process allows both linear and rotational work to be done in the generation of electricity and the movement of vehicles. The initial understanding of transient heat transfer was developed by Newton[10] and started with Newton's law of cooling:[11]

$$\frac{dQ}{dt} = h^*A^*(T_S - T_E) \tag{6.3}$$

where $\frac{dQ}{dt}$ = incremental energy lost in joules per unit time (J/s)
A = energy transmission surface area (m^2)
h = heat transfer coefficient [$W/(m^2*K]$
T_S = surface temperature of the object losing heat (K)
T_E = temperature of the environment gaining heat (K)

Subsequent work by Jean Baptiste Joseph Fourier,[12] based on Newton's law of cooling, developed the law for steady-state heat conduction (known as Fourier's law[13]). Fourier's law is expressed here in differential form:

$$q = -k\nabla T \tag{6.4}$$

where q = heat flux in watts per square meter (W/m^2)
k = thermal conductivity of the material [$W/(m^*K]$
∇T = temperature gradient (K/m)

2D Resistive Heating Modeling

2D Resistive Heating Model

The following numerical solution model (Resistive_Heating_1) is derived from a model that was originally developed by COMSOL as a Multiphysics demonstration model for distribution with the Multiphysics software in the basic Multiphysics Model Library. This model introduces the coupling of two important basic Application Modes: Joule Heating in the Conductive Media DC Application Mode and the Heat Transfer by Conduction Application Mode. The coupling of these two modes in this model demonstrates the interactions normally found in typical engineering materials.

NOTE It is important for the new modeler to personally build each model presented in this text. There is no substitute in the path to understanding of the modeling process for the hands-on experience of actually building, meshing, solving, and postprocessing a model. Many times the inexperienced modeler will make and subsequently correct errors, there by adding to his or her experience and fund of modeling knowledge. Even building the simplest model will expand the modeler's store of knowledge.

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat generation during a process and either add heat or remove heat to achieve or maintain a desired temperature. Figure 6.3 shows a 3D rendition of the 2D resistive heating geometry, as will be modeled in this section.

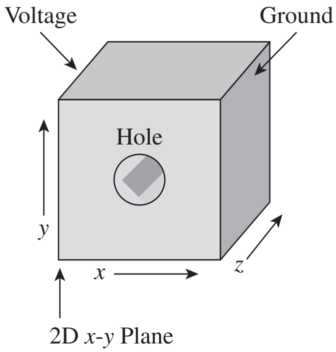


FIGURE 6.3 3D rendition of the 2D resistive heating model

To start building the Resistive_Heating_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (default setting) from the Space dimension pull-down list. Select COMSOL Multiphysics > Electro-Thermal Interaction > Joule Heating > Transient analysis. See Figure 6.4. Click OK.

FIGURE 6.4 2D Resistive_Heating_1 Model Navigator setup

Table 6.1 Constants Edit Window

Name	Expression	Description
r_Cu	1.754e-8[ohm*m]	Resistivity of copper at T_ref
T_ref	20[degC]	Reference temperature
alpha_Cu	3.9e-3[1/K]	Temperature coefficient copper
V_0	1e-1[V]	Electric potential (voltage)
T_air	300[K]	Air temperature
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity copper
rho_Cu	8.96e3[kg/m^3]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity copper

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 6.1; also see Figure 6.5. Click OK.

NOTE When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 1.0 and a height of 1.0. Select "Base: Center" and set x equal to 0 and y equal to 0 in the Rectangle edit window. See Figure 6.6.

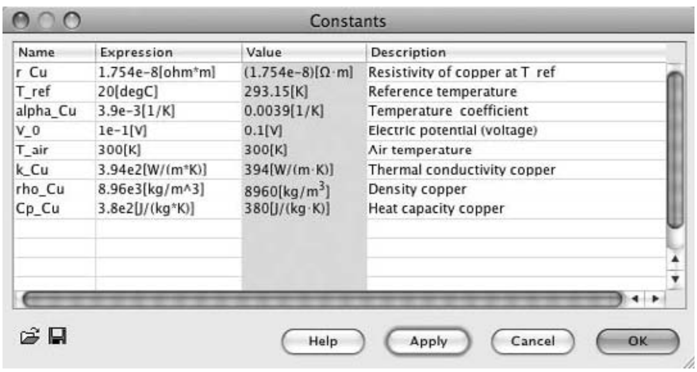


FIGURE 6.5 2D Resistive_Heating_1 model Constants edit window

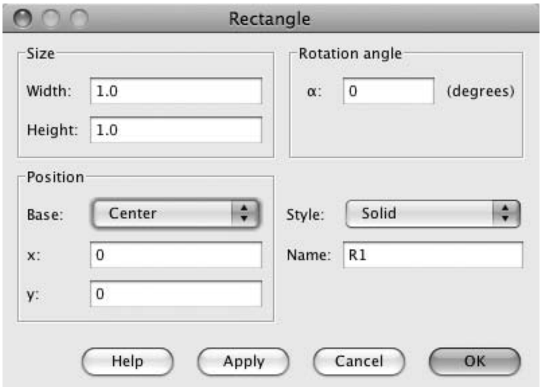


FIGURE 6.6 2D Resistive_Heating_1 model Rectangle edit window

Click OK, and then click the Zoom Extents button. See Figure 6.7.

Using the menu bar, select Draw > Specify Objects > Circle. In the Circle edit window, enter a radius of 0.1 and a base of "Center." Set x equal to 0 and y equal to 0. See Figure 6.8.

Click OK. See Figure 6.9.

FIGURE 6.7 2D Resistive_Heating_1 model rectangle

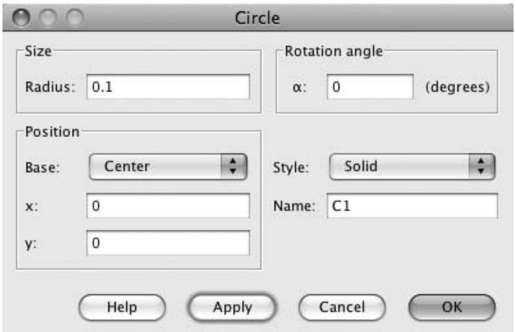


FIGURE 6.8 2D Resistive_Heating_1 model Circle edit window

NOTE The rectangle is the 2D representation of a cube in cross section. The circle is added to the 2D geometry to allow the creation of a hole through the cube, as shown in Figure 6.3.

Using the menu bar, select Draw > Create Composite Object. In the Set formula edit window, enter R1−C1. See Figure 6.10.

FIGURE 6.9 2D Resistive_Heating_1 model rectangle and circle

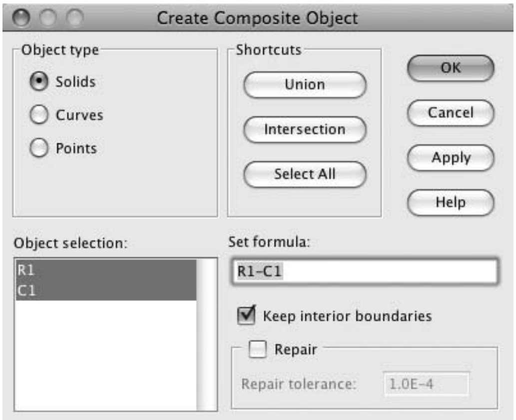


FIGURE 6.10 2D Resistive_Heating_1 model Create Composite Object edit window

Click OK. See Figure 6.11.

This model introduces the coupling of two basic Application Modes: Joule Heating in the Conductive Media DC Application Mode and the Heat Transfer by Conduction Application Mode. The Physics subdomain and boundary settings will

FIGURE 6.11 2D Resistive_Heating_1 model, block with hole

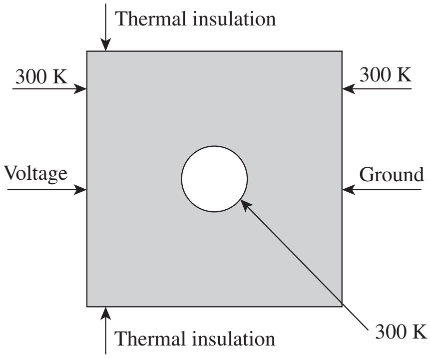


FIGURE 6.12 2D Resistive_Heating_1 model boundary conditions overview

need to be specified in each mode separately. Figure 6.12 shows an overview of the boundary conditions for the combination of both modes.

First, however, the subdomain settings values need to be specified.

Physics Subdomain Settings: Heat Transfer by Conduction (ht)

Having established the geometry for the 2D Resistive_Heating_1 model of a block with a hole, the next step is to define the fundamental Physics conditions. Using the menu bar, select Multiphysics > Heat Transfer by Conduction (ht).

Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). In the Subdomain edit windows, enter the information shown in Table 6.2. See Figure 6.13.

NOTE For transient calculations, all of the physical property values are required for the conduction calculation. If *C*p and rho are set to zero, the implication is that the material is a perfect vacuum, which is logically inconsistent with the stated value of *k*.

Select the Init tab. Enter T_ref in the Initial value edit window. See Figure 6.14. Click OK.

Table 6.2 Subdomain Edit Window

Name	Expression	Description
k (isotropic)	k_Cu	Thermal conductivity
ρ	rho_Cu	Density
C_P	*C*p_Cu	Heat capacity

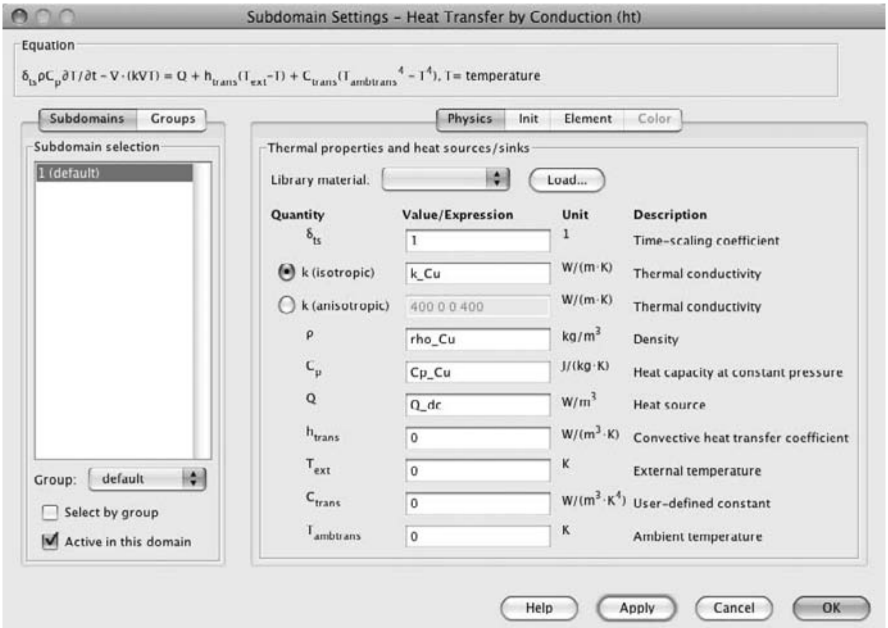


FIGURE 6.13 2D Resistive_Heating_1 model Subdomain Settings edit window

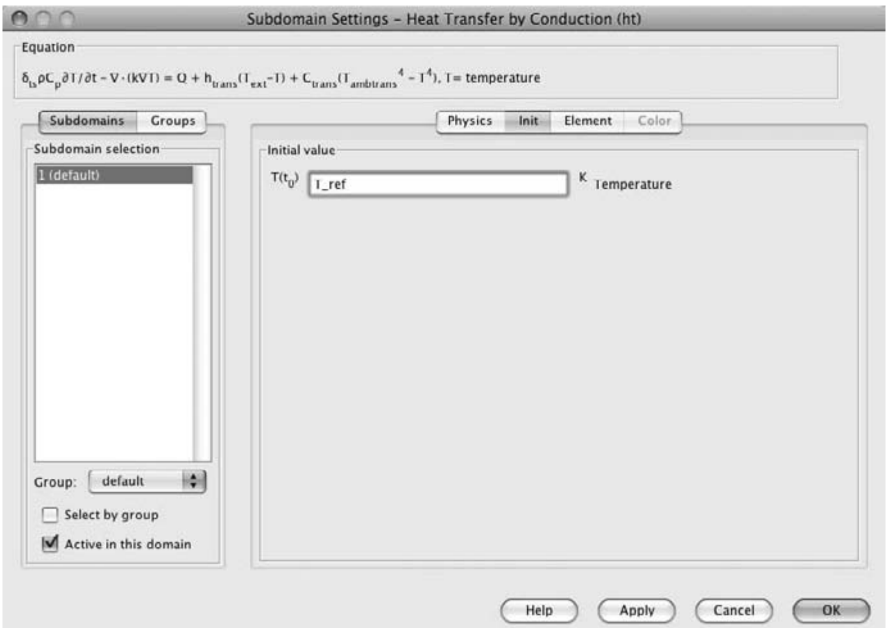


FIGURE 6.14 2D Resistive_Heating_1 model Subdomain Settings, Init edit window

Table 6.3 Boundary Settings–Heat Transfer by Conduction (ht) Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
1, 4–8	Temperature	T_air	6.15
2, 3	Thermal insulation	—	6.16

Physics Boundary Settings: Heat Transfer by Conduction (ht)

Using the menu bar, select Physics > Boundary Setting. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 6.3. Click OK. See Figures 6.15 and 6.16.

Physics Subdomain Settings: Conductive Media DC (dc)

Using the menu bar, in the Model Navigator menu, select Multiphysics > Conductive Media DC (dc). Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). Select "Linear temperature relation" from the Conductivity relation pull-down list. In the Subdomain edit windows, enter the information as shown in Table 6.4. See Figure 6.17.

NOTE At this point in the model, the generation of heat is coupled to the resistivity through the temperature change.

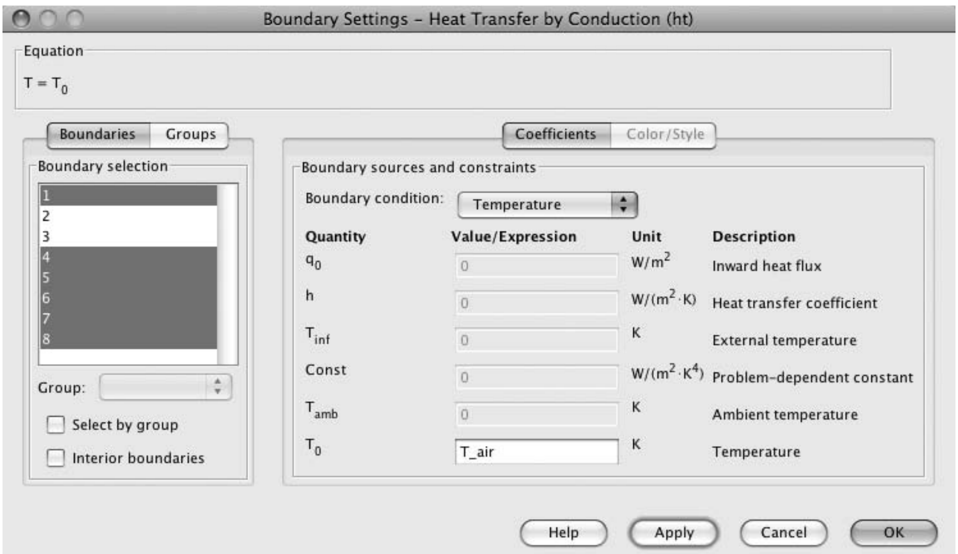


FIGURE 6.15 2D Resistive_Heating_1 model Boundary Settings (1, 4–8) edit window

FIGURE 6.16 2D Resistive_Heating_1 model Boundary Settings (2, 3) edit window

Select the Init tab. Enter V_0*(1−x[1/m]) in the $V(t_0)$ edit window. See Figure 6.18. Click OK.

NOTE The initial conditions assume a linear voltage drop across the body of the model. This is reflected in the initialization equation ($V(t_0)$ = V_0*(1−x[1/m])).

Physics Boundary Settings: Conductive Media DC (dc)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 6.5. Click OK. See Figures 6.19, 6.20, and 6.21.

Table 6.4 Subdomain Settings–Conductive Media DC (dc) Edit Window

Name	Expression	Description
ρ_0	r_Cu	Resistivity at reference temperature
α	alpha_Cu	Temperature coefficient
T_0	T_ref	Reference temperature

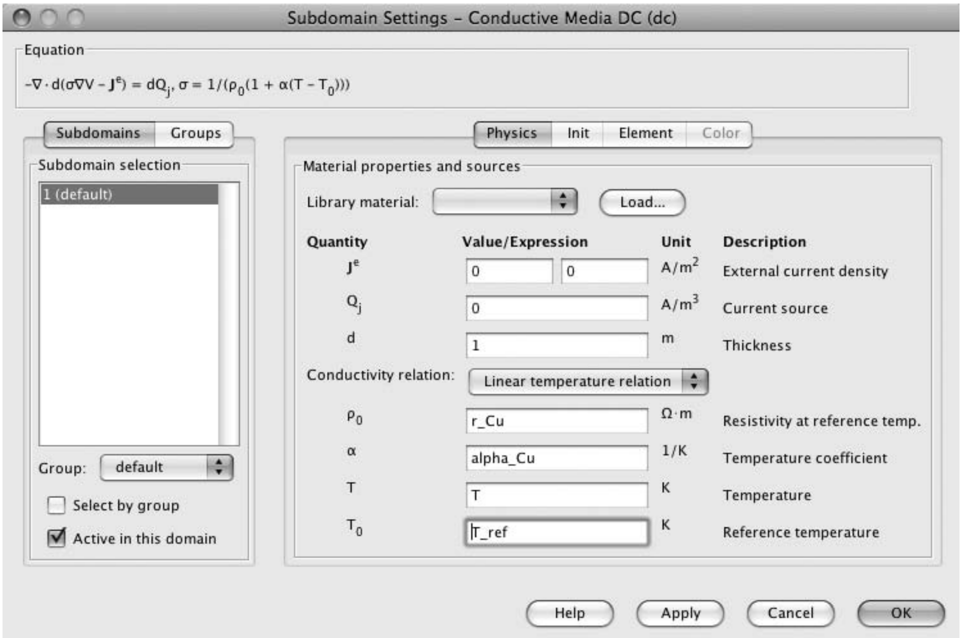


FIGURE 6.17 2D Resistive_Heating_1 model Subdomain Settings edit window

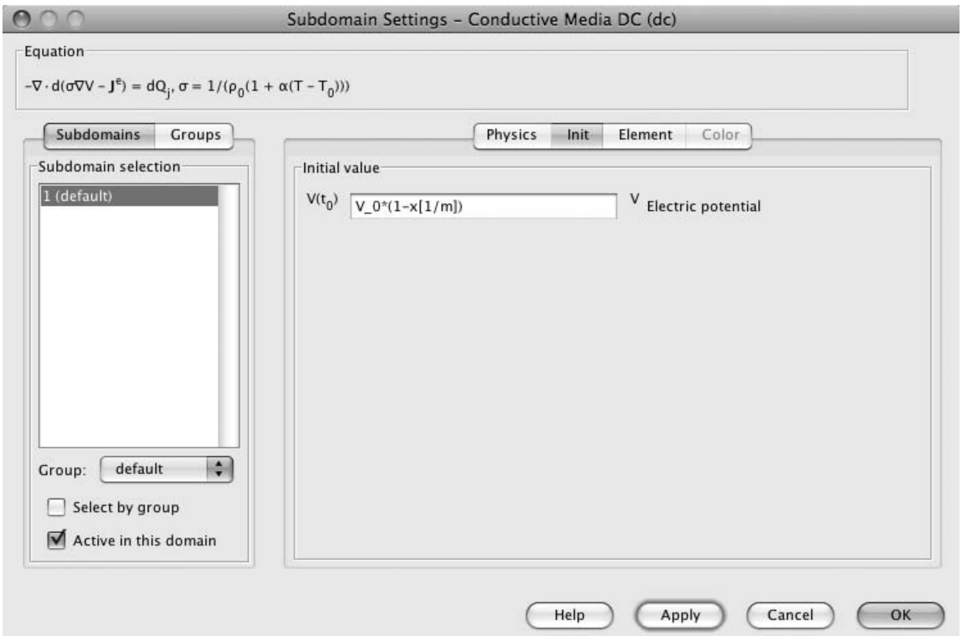


FIGURE 6.18 2D Resistive_Heating_1 model Subdomain Settings Init edit window

Table 6.5 Boundary Settings–Conductive Media DC (dc) Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
1	Electric potential	V_0	6.19
2, 3, 5–8	Electric insulation	—	6.20
4	Ground	—	6.21

FIGURE 6.19 2D Resistive_Heating_1 model Boundary Settings (1) edit window

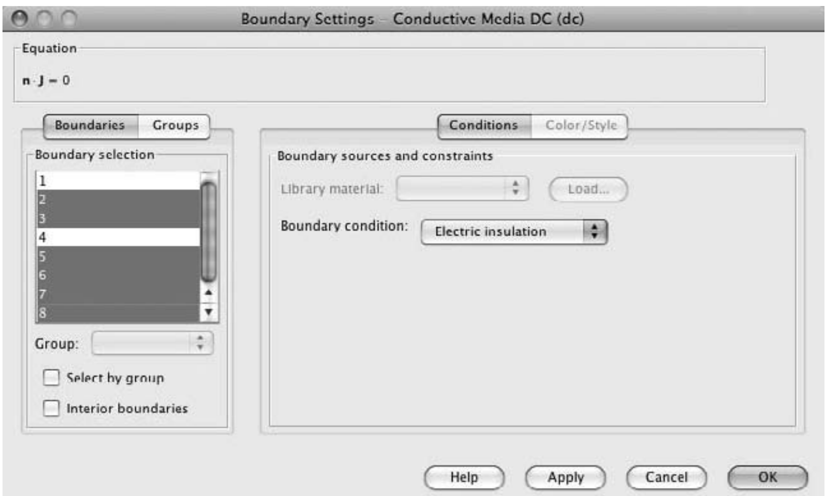


FIGURE 6.20 2D Resistive_Heating_1 model Boundary Settings (2, 3, 5–8) edit window

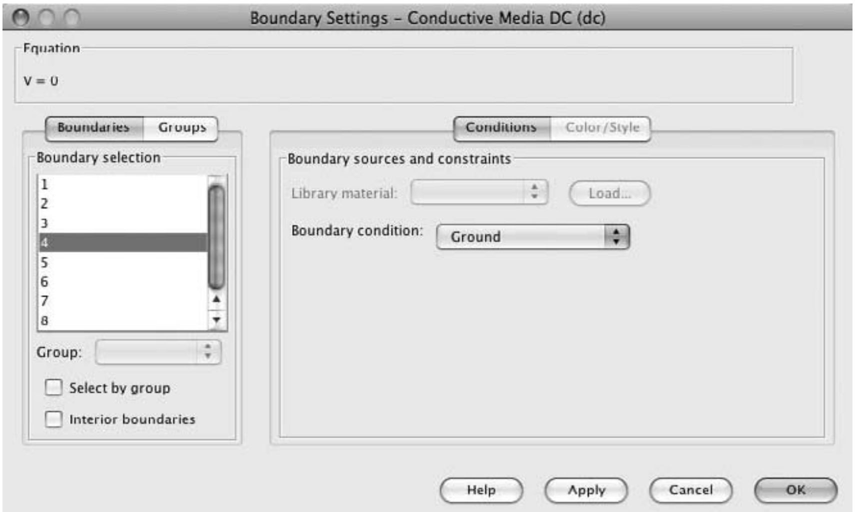


FIGURE 6.21 2D Resistive_Heating_1 model Boundary Settings (4) edit window

Figure 6.22 shows the 2D Resistive_Heating_1 model with all the boundary settings.

Mesh Generation

On the toolbar, click the Initialize Mesh button once. Click the Refine Mesh button once. This results in a mesh of approximately 4300 elements. See Figure 6.23.

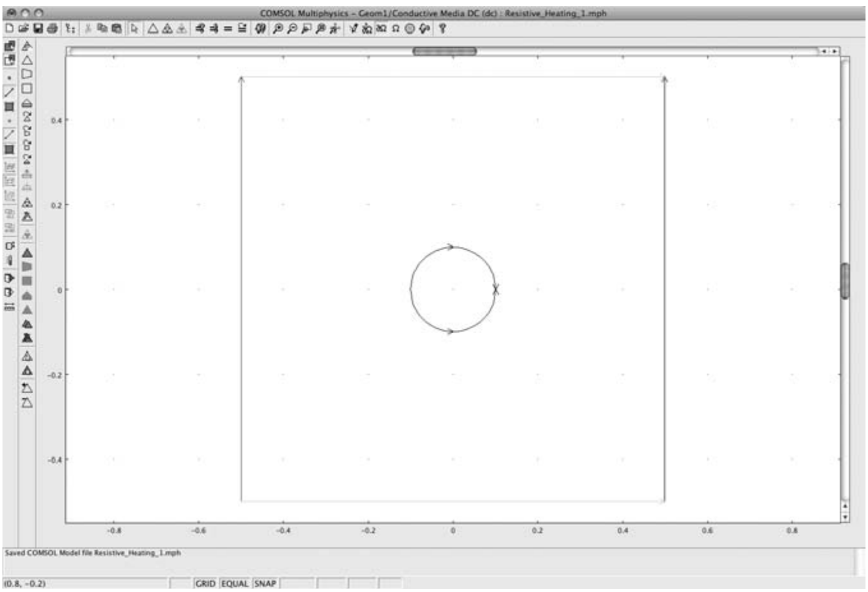


FIGURE 6.22 2D Resistive_Heating_1 model with all the boundary settings

FIGURE 6.23 2D Resistive_Heating_1 model mesh window

Solving the 2D Resistive_Heating_1 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Enter 0:50:2000 in the Times edit window. See Figure 6.24. Click OK.

Time-Dependent Solving of the 2D Resistive_Heating_1 Model

Select Solve > Solve Problem. See Figure 6.25.

Postprocessing and Visualization

The default plot shows the temperature distribution in Kelvin. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot

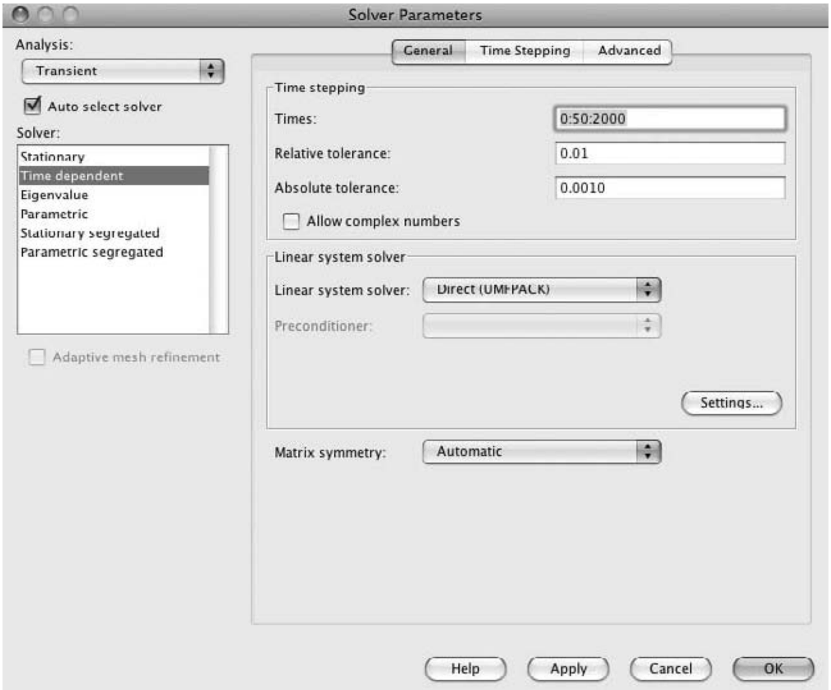


FIGURE 6.24 2D Resistive_Heating_1 model Solver Parameters edit window

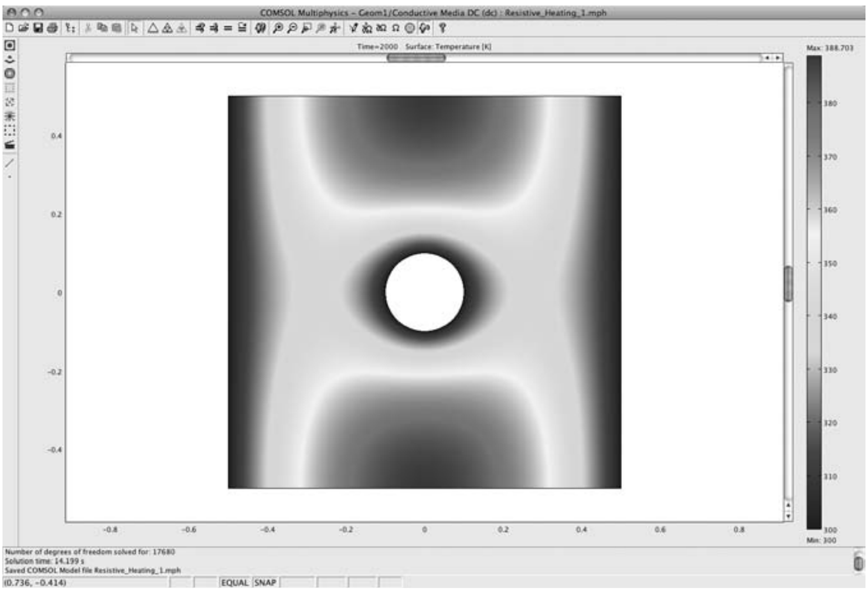


FIGURE 6.25 2D Resistive_Heating_1 model solution

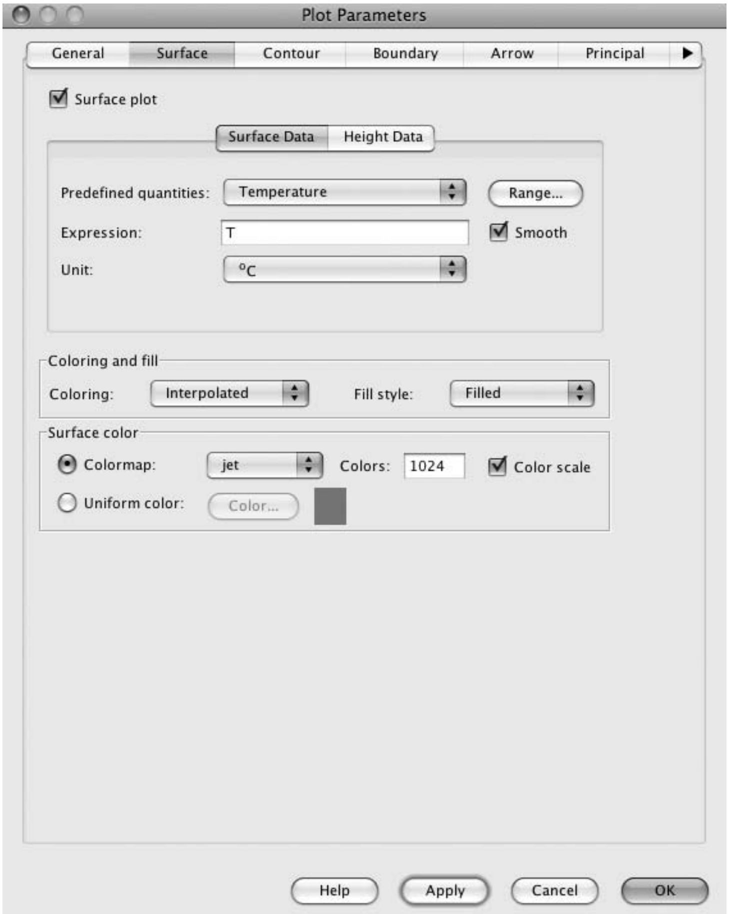


FIGURE 6.26 2D Resistive_Heating_1 model Plot Parameters edit window

Parameters > Surface. Verify that the Surface plot check box is checked and that the Predefined quantities pulldown list shows "Temperature." Select "degC" or "°C" from the Unit pull-down list. See Figure 6.26.

Click OK. See Figure 6.27.

It is relatively simple to demonstrate the heat flux. Select Postprocessing > Plot Parameters > Arrow. Check the Arrow plot check box. Select Heat Transfer by Conduction (ht) > Heat flux from the Predefined quantities pull-down list. Click the Color button and select a color such as "black." Click OK. See Figure 6.28.

Click OK. See Figure 6.29.

Next, because this is a transient analysis model, the modeler can test how close the solution is to the steady-state value. Select Postprocessing > Cross-Section Plot Parameters > General > Point plot. Verify that all of the Solutions to use are selected. See Figure 6.30.

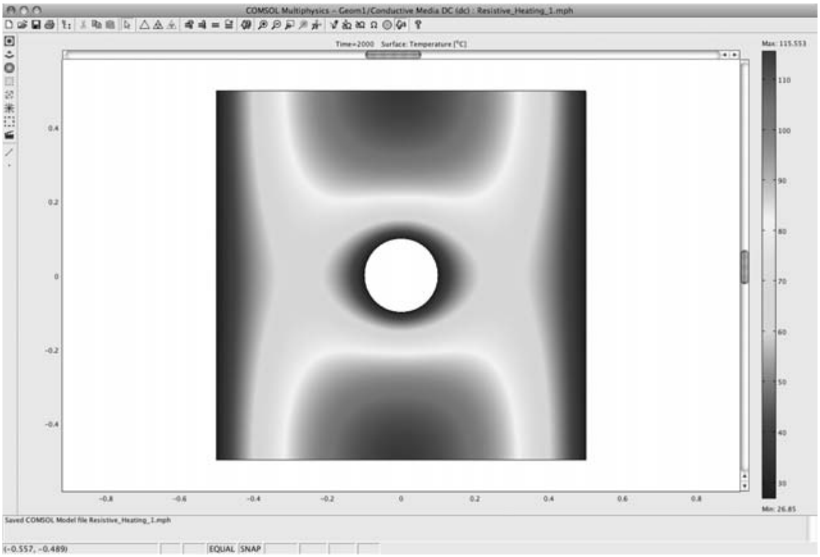


FIGURE 6.27 2D Resistive_Heating_1 model, degrees Centigrade

FIGURE 6.28 2D Resistive_Heating_1 model, Plot Parameters, Arrow edit window

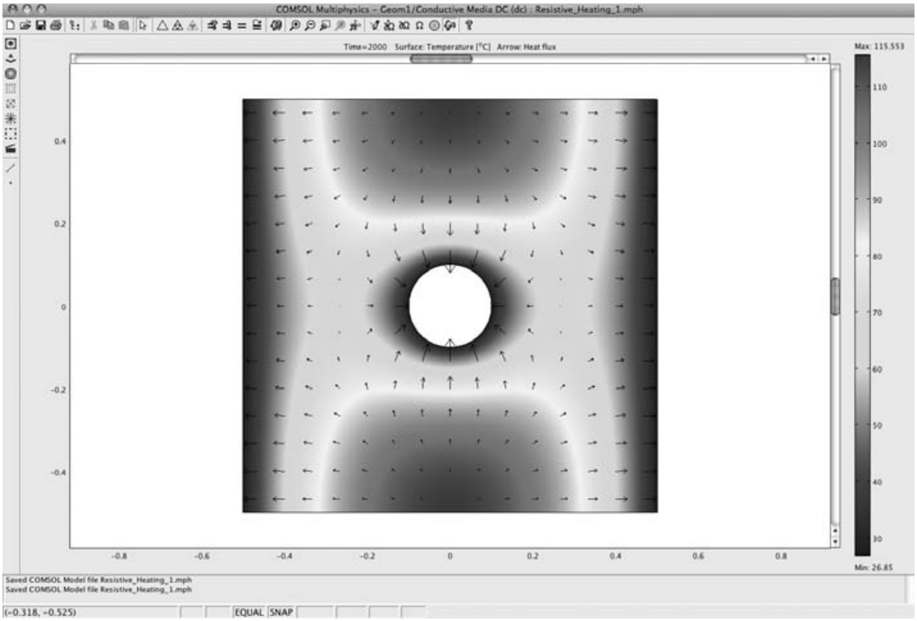


FIGURE 6.29 2D Resistive_Heating_1 model, temperature and heat flux

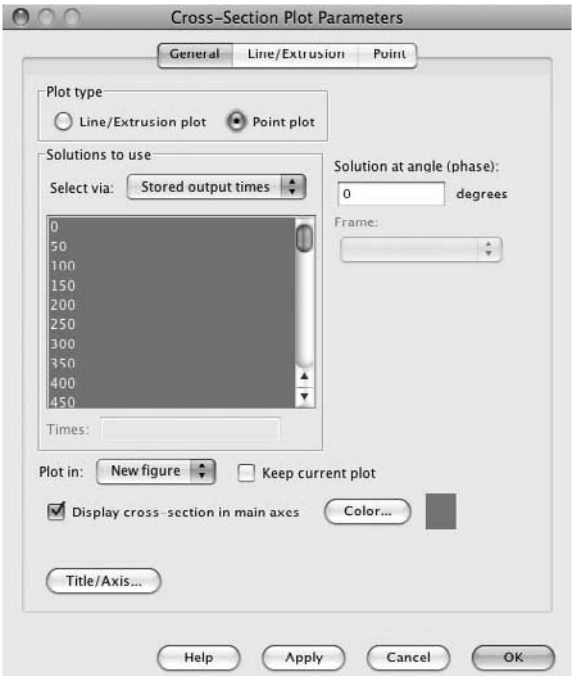


FIGURE 6.30 2D Resistive_Heating_1 model, Cross-Section Plot Parameters, General edit window

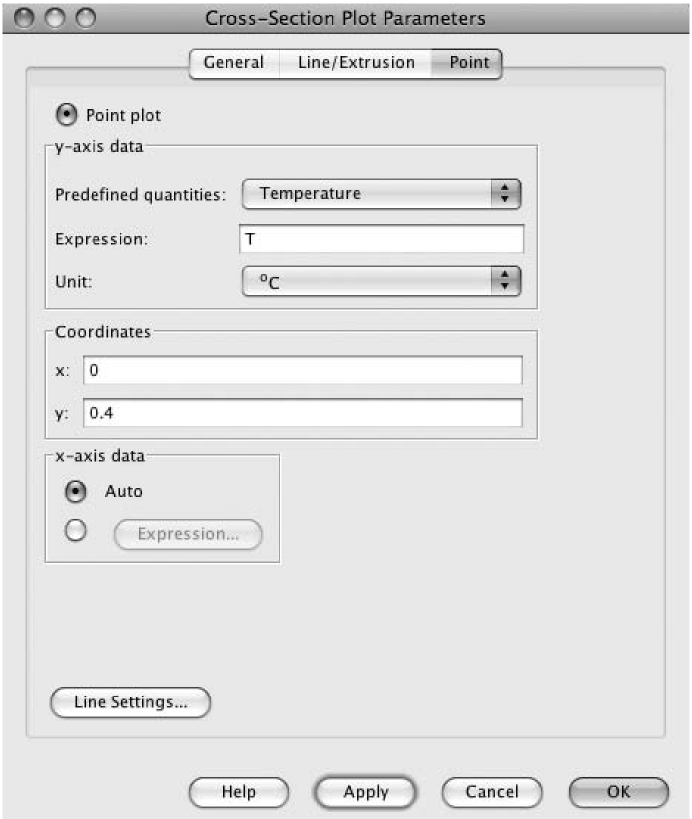


FIGURE 6.31 2D Resistive_Heating_1 model, Cross-Section Plot Parameters, Point edit window

Click the Point tab. Select "°C" from the Unit pull-down list. Enter x = 0, y = 0.4 in the Coordinates edit windows. See Figure 6.31.

Click OK. Figure 6.32 shows the temperature versus time plot for the point $x = 0$, $y = 0.4$. It is easily seen that the temperature is close to the steady-state value (the curve approaches the horizontal, small ΔT) at the end of the modeling calculation.

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.33.

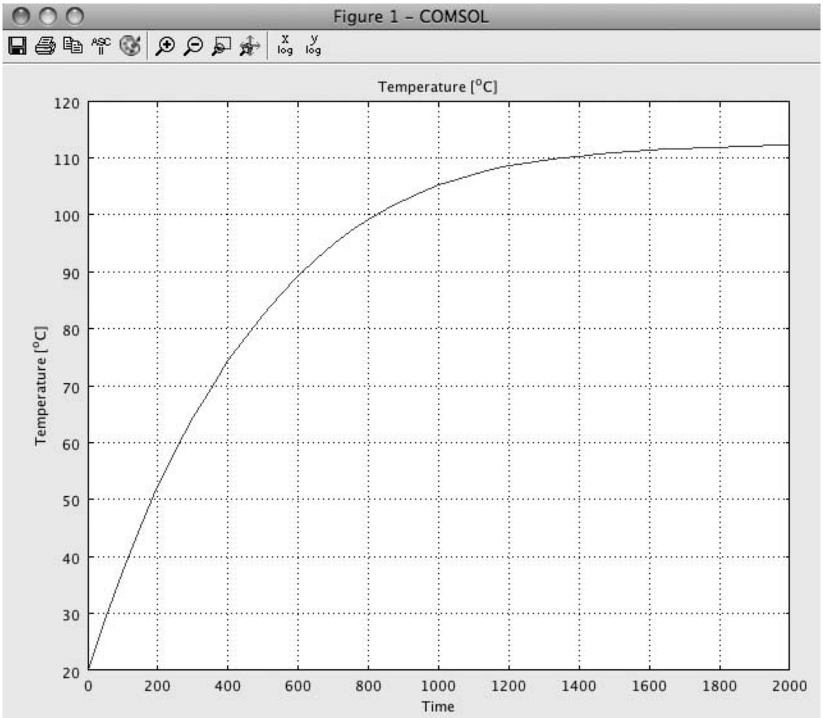


FIGURE 6.32 2D Resistive_Heating_1 model, temperature versus time at $x = 0$, $y = 0.4$

Click the Start Animation button. See Figure 6.34.

Alternatively, you can play the file Movie6_RH_1.avi that was supplied with this book.

First Variation on the 2D Resistive Heating Model

The following numerical solution model (Resistive_Heating_2) is derived from the model Resistive_Heating_1. In this model, geometric and materials composition changes are introduced, such as might be used in a general industrial application. It is a multielement heating unit with Nichrome (a nickel–chromium alloy) heating bars and copper connecting bars.

NOTE The Resistive_Heating_2 model demonstrates materials and a configuration as might be employed in heat sealers, soldering heads, packaging equipment, and printed circuit board processing equipment.

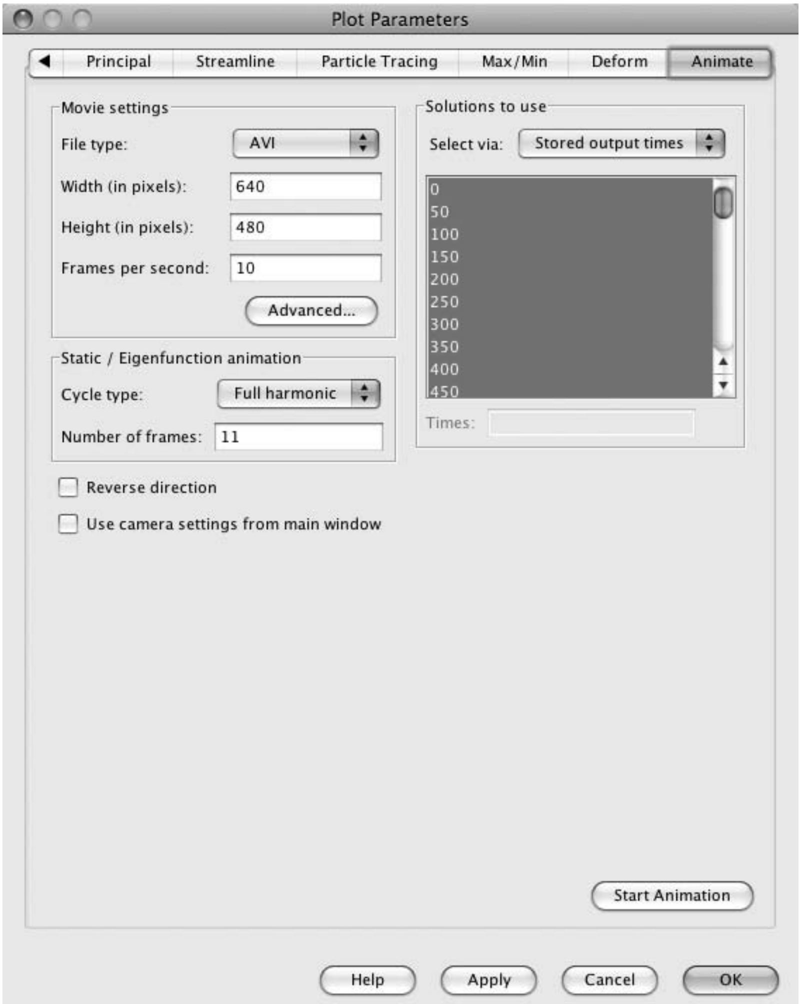


FIGURE 6.33 2D Resistive_Heating_1 model, Plot Parameters window

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat generation during a process and either add heat or remove heat so as to achieve or maintain a desired temperature. Figure 6.35 shows a 3D rendition of the 2D resistive heating geometry, as will be modeled here.

To start building the Resistive_Heating_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from

FIGURE 6.34 2D Resistive_Heating_1 model animation, final frame

the Space dimension pull-down list. Select COMSOL Multiphysics > Electro-Thermal Interaction > Joule Heating > Transient analysis. See Figure 6.36. Click OK.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 6.6; see also Figure 6.37. Click OK.

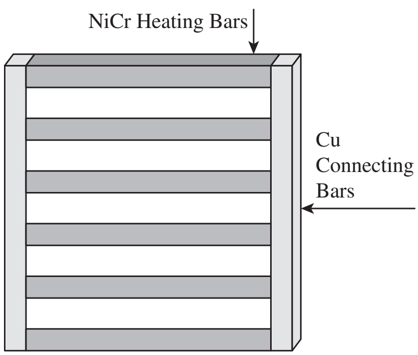


FIGURE 6.35 3D rendition of the 2D Resistive_Heating_2 model (not to scale)

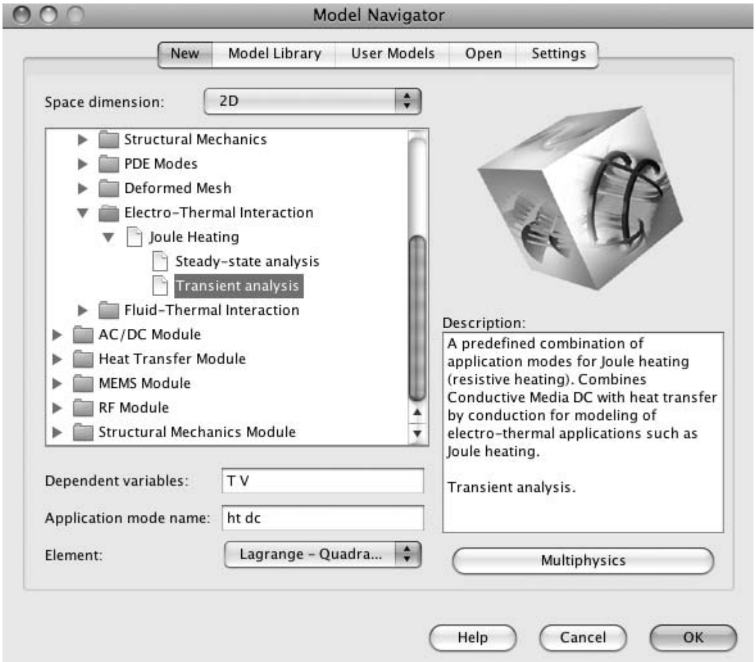
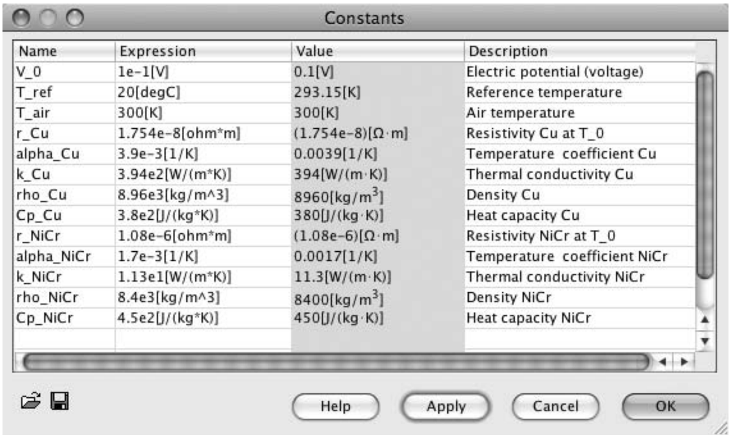


FIGURE 6.36 2D Resistive_Heating_2 Model Navigator setup

Table 6.6 Constants Edit Window

Name	Expression	Description
V_0	1 [V]	Electric potential (voltage)
T_ref	20[degC]	Reference temperature
T_air	300[K]	Air temperature
r_Cu	1.754e-8[ohm*m]	Resistivity Cu at T_0
alpha_Cu	3.9e-3[1/K]	Temperature coefficient Cu
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity Cu
rho_Cu	8.96e3[kg/m^3]	Density Cu
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity Cu
r_NiCr	1.08e-6[ohm*m]	Resistivity NiCr at T_0
alpha_NiCr	1.7e-3[1/K]	Temperature coefficient NiCr
k_NiCr	1.13e1[W/(m*K)]	Thermal conductivity NiCr
rho_NiCr	8.4e3[kg/m^3]	Density NiCr
Cp_NiCr	4.5e2[J/(kg*K)]	Heat capacity NiCr



Name	Expression	Value	Description
V_0	1e-1[V]	0.1[V]	Electric potential (voltage)
T_ref	20[degC]	293.15[K]	Reference temperature
T_air	300[K]	300[K]	Air temperature
r_Cu	1.754e-8[ohm*m]	(1.754e-8)[Ω·m]	Resistivity Cu at T_0
alpha_Cu	3.9e-3[1/K]	0.0039[1/K]	Temperature coefficient Cu
k_Cu	3.94e2[W/(m*K)]	394[W/(m·K)]	Thermal conductivity Cu
rho_Cu	8.96e3[kg/m^3]	8960[kg/m³]	Density Cu
Cp_Cu	3.8e2[J/(kg*K)]	380[J/(kg·K)]	Heat capacity Cu
r_NiCr	1.08e-6[ohm*m]	(1.08e-6)[Ω·m]	Resistivity NiCr at T_0
alpha_NiCr	1.7e-3[1/K]	0.0017[1/K]	Temperature coefficient NiCr
k_NiCr	1.13e1[W/(m*K)]	11.3[W/(m·K)]	Thermal conductivity NiCr
rho_NiCr	8.4e3[kg/m^3]	8400[kg/m³]	Density NiCr
Cp_NiCr	4.5e2[J/(kg*K)]	450[J/(kg·K)]	Heat capacity NiCr

FIGURE 6.37 2D Resistive_Heating_2 model Constants edit window

NOTE In building this model, the calculational parameters (e.g., constants, scalar expressions) have been consolidated into a convenient location (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Draw > Specify Objects > Rectangle. Create each of the rectangles indicated in Table 6.7. See Figure 6.38.

Using the menu bar, select Draw > Create Composite Object. Enter: R1+R7+R8−R2−R3−R4−R5−R6. See Figure 6.39.

Table 6.7 Rectangle Edit Window

R Number	Width	Height	Base	*x*	*y*
1	1.0	1.1	Center	0	0
2	0.9	0.1	Corner	−0.45	0.35
3	0.9	0.1	Corner	−0.45	0.15
4	0.9	0.1	Corner	−0.45	−0.05
5	0.9	0.1	Corner	−0.45	−0.25
6	0.9	0.1	Corner	−0.45	−0.45
7	0.05	1.1	Corner	−0.50	−0.55
8	0.05	1.1	Corner	−0.45	−0.55

FIGURE 6.38 2D Resistive_Heating_2 model created rectangles

Click OK, and then click the Zoom Extents button. See Figure 6.40.

NOTE In building this model, the same geometry has been built that will be used in the next model. To save the modeler some time, select File > Export > Geometry Objects to File. Enter RH2_Geometry in the Save As edit window. Click the Save button.

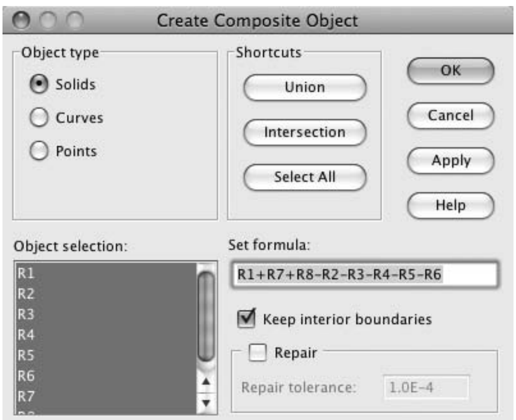


FIGURE 6.39 2D Resistive_Heating_2 model Create Composite Object edit window

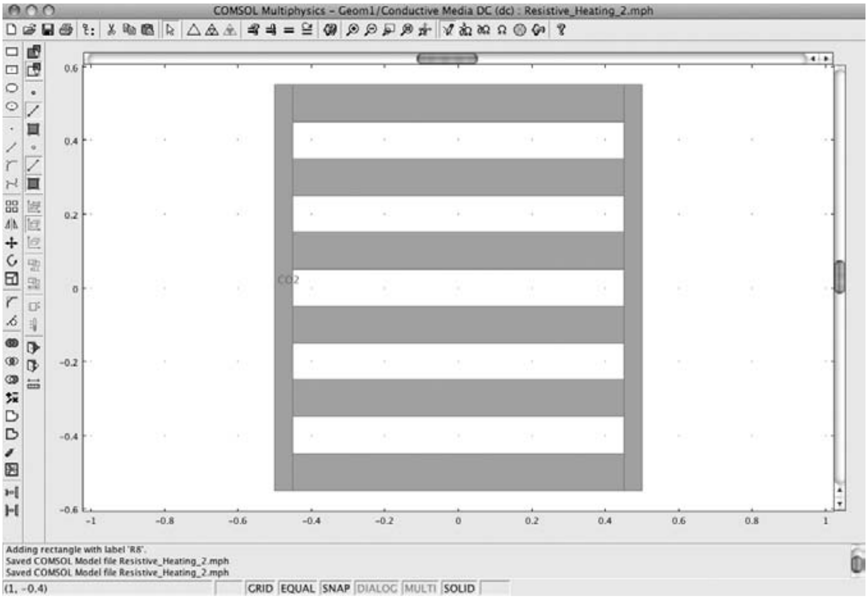


FIGURE 6.40 2D Resistive_Heating_2 model heater bar assembly

This model introduces the coupling of two basic Application Modes: Joule Heating in the Conductive Media DC Application Mode and the Heat Transfer by Conduction Application Mode. The Physics subdomain and boundary settings will need to be specified in each mode separately. Figure 6.41 shows an overview of the boundary conditions for the combination of both modes.

First, however, the subdomain settings values need to be specified.

FIGURE 6.41 2D Resistive_Heating_2 model boundary conditions overview (not to scale)

Table 6.8 Subdomain Settings Edit Window

Subdomain Number	Name	Expression	Description
1, 8	k (isotropic)	k_Cu	Thermal conductivity
	ρ	rho_Cu	Density
	C_P	Cp_Cu	Heat capacity

Physics Subdomain Settings: Heat Transfer by Conduction (ht)

Having established the geometry for the 2D Resistive_Heating_2 model of a heater bar assembly, the next step is to define the fundamental Physics conditions. Using the menu bar, select Multiphysics > Heat Transfer by Conduction (ht). Using the menu bar, select Physics > Subdomain Settings. In the Subdomain edit windows, enter the information shown in Table 6.8. Click the Apply button. See Figure 6.42.

In the Subdomain edit windows, enter the information shown in Table 6.9. Click the Apply button. See Figure 6.43.

FIGURE 6.42 2D Resistive_Heating_2 model Subdomain Settings (1, 8) edit window

Table 6.9 Subdomain Settings Edit Window

Subdomain Number	Name	Expression	Description
2–7	*k* (isotropic)	k_NiCr	Thermal conductivity
	ρ	rho_NiCr	Density
	C_p	Cp_NiCr	Heat capacity

NOTE For transient calculations, all of the physical property values are required for the conduction calculation. In this case, the properties of both copper (Cu) and Nichrome (NiCr) are required.

Select the Init tab. Select subdomains 1–8. Enter T_ref in the Initial value edit window. See Figure 6.44. Click OK.

Physics Boundary Settings: Heat Transfer by Conduction (ht)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 6.10. See Figures 6.45 and 6.46.

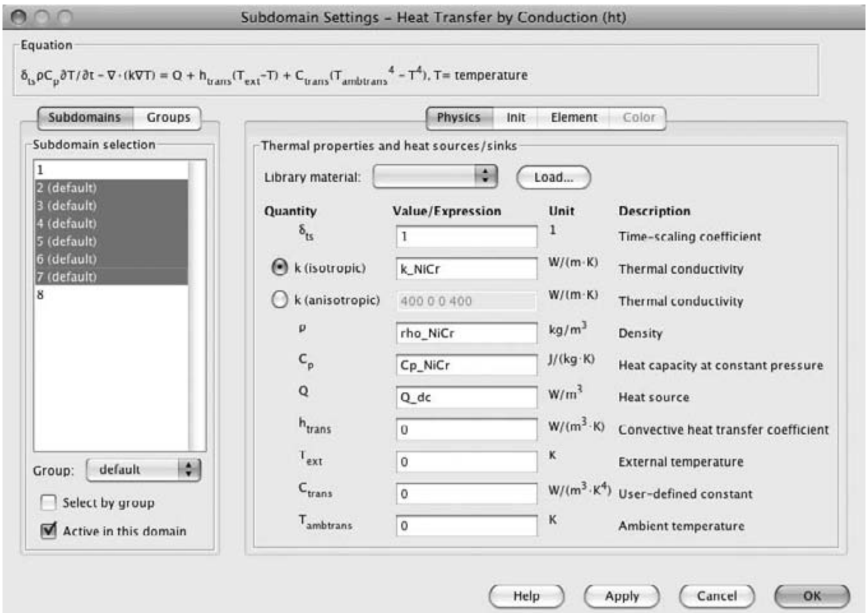


FIGURE 6.43 2D Resistive_Heating_2 model Subdomain Settings (2–7) edit window

FIGURE 6.44 2D Resistive_Heating_2 model Subdomain Settings, Init edit window

Click OK. Figure 6.47 shows the final combined Heat Transfer by Conduction (ht) boundary settings.

Physics Subdomain Settings: Conductive Media DC (dc)

Using the menu bar, select Multiphysics > Conductive Media DC (dc). Using the menu bar, select Physics > Subdomain Settings. Select Subdomains 1–8 in the Subdomain

Table 6.10 Boundary Settings–Heat Transfer by Conduction (ht) Edit Window

Boundary	Boundary Condition	Value/ Expression	Click Apply	Figure Number
1, 40	Temperature	T_air	Yes	6.45
2, 3, 5–7, 9–11, 13–15, 17–19, 21–23, 25, 26, 28, 29, 31, 33, 35, 37, 39	Thermal insulation	—	Yes	6.46

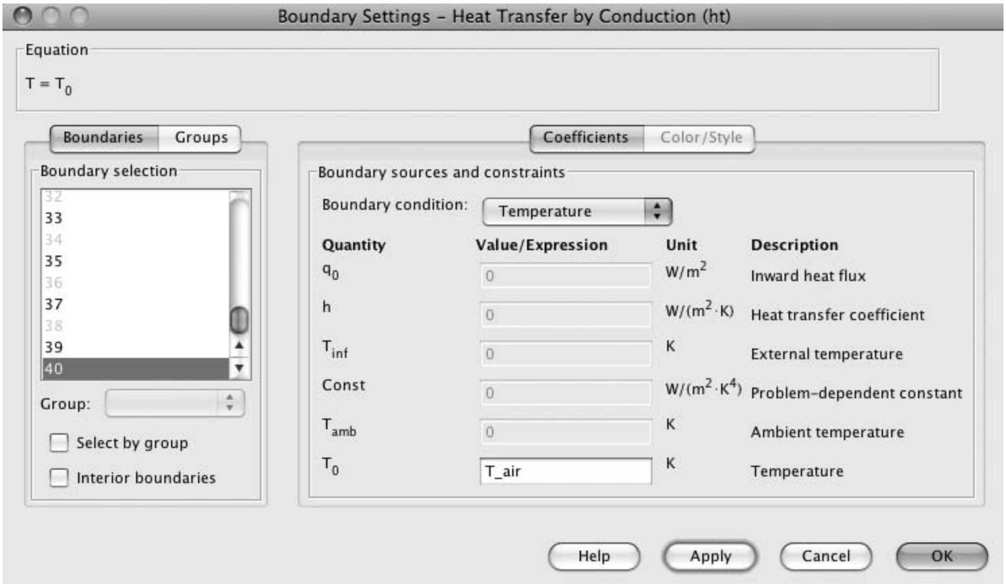


FIGURE 6.45 2D Resistive_Heating_2 model Boundary Settings (1, 40) edit window

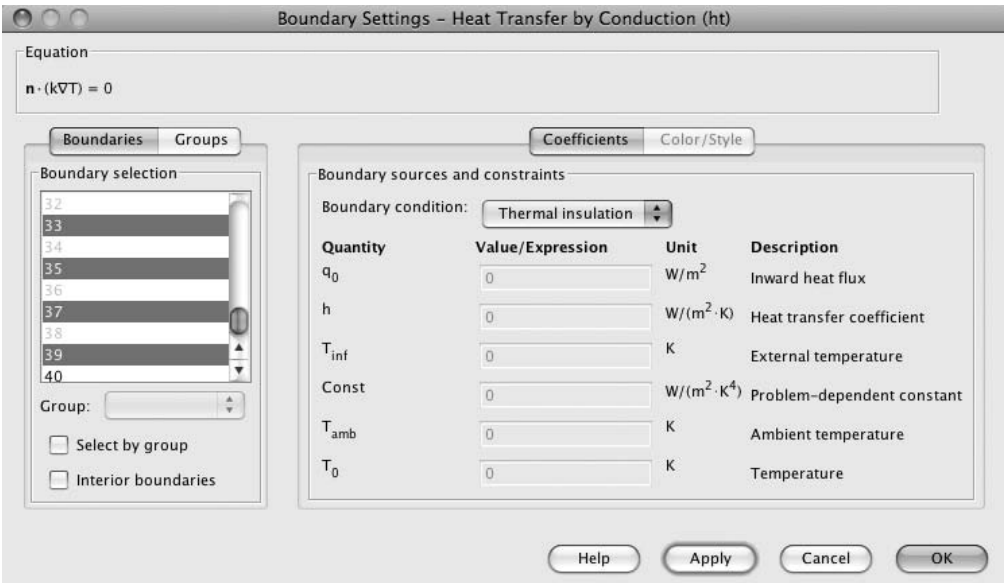


FIGURE 6.46 2D Resistive_Heating_2 model Boundary Settings (2, 3…) edit window

FIGURE 6.47 2D Resistive_Heating_2 model Combined Heat Transfer by Conduction (ht) boundary settings

selection window (all of the subdomains). Enter 0.01 in the Thickness (d) edit window. Select "Linear temperature relation" from the Conductivity relation pull-down list. Click the Apply button.

In the Subdomain edit windows, enter the information shown in Table 6.11. Click the Apply button. See Figure 6.48.

In the Subdomain edit windows, enter the information shown in Table 6.12. Click the Apply button. See Figure 6.49.

NOTE At this point in the model, the generation of heat is coupled to the resistivity in each different material (Cu, NiCr) through the temperature change.

Table 6.11 Subdomain Settings–Conductive Media DC (dc) Edit Window

Subdomain Number	Name	Expression	Description
1, 8	ρ_0	r_Cu	Resistivity at reference temperature
	α	alpha_Cu	Temperature coefficient
	T_0	T_ref	Reference temperature

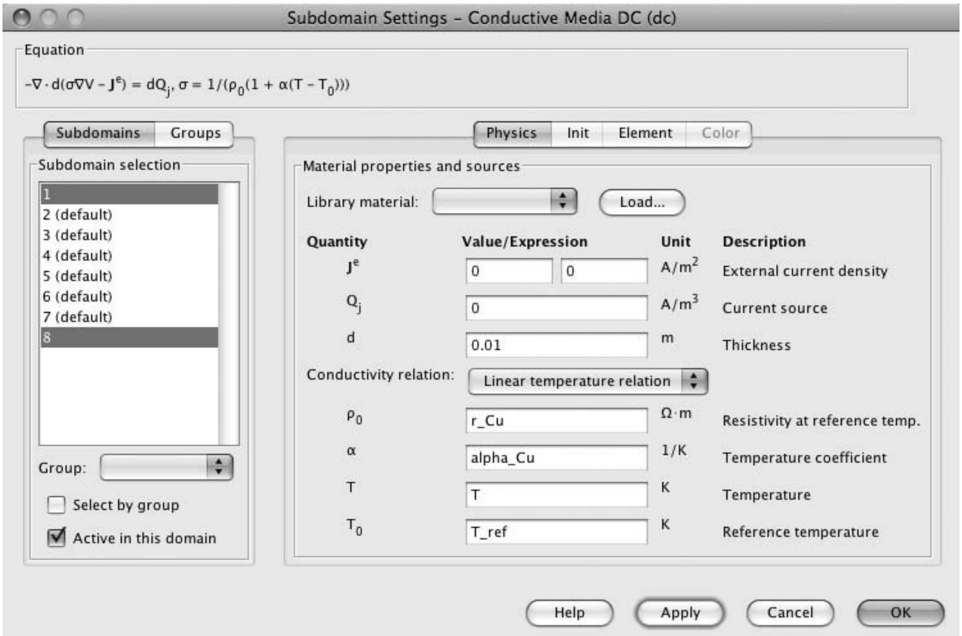


FIGURE 6.48 2D Resistive_Heating_2 model Subdomain Settings (1, 8) edit window

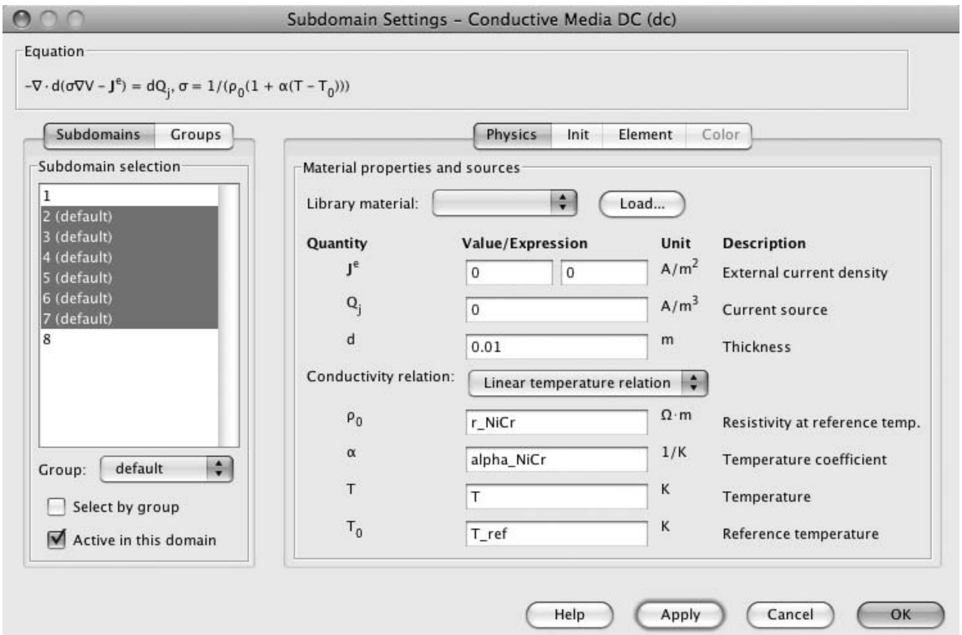


FIGURE 6.49 2D Resistive_Heating_2 model Subdomain Settings (2–7) edit window

Table 6.12 Subdomain Settings–Conductive Media DC (dc) Edit Window

Subdomain Number	Name	Expression	Description
2–7	ρ_0	r_NiCr	Resistivity at reference temperature
	α	alpha_NiCr	Temperature coefficient
	T_0	T_ref	Reference temperature

Select the Init tab. Select subdomains 1–8 in the Subdomain selection window (all of the subdomains). Enter V_0*(1 − x[1/m]) in the V(t_0) edit window. See Figure 6.50. Click OK.

NOTE The initial conditions assume a linear voltage drop across the body of the model. This is reflected in the initialization equation (V(t_0) = V_0*(1 − x[1/m])).

Physics Boundary Settings: Conductive Media DC (dc)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 6.13. See Figures 6.51, 6.52, and 6.53.

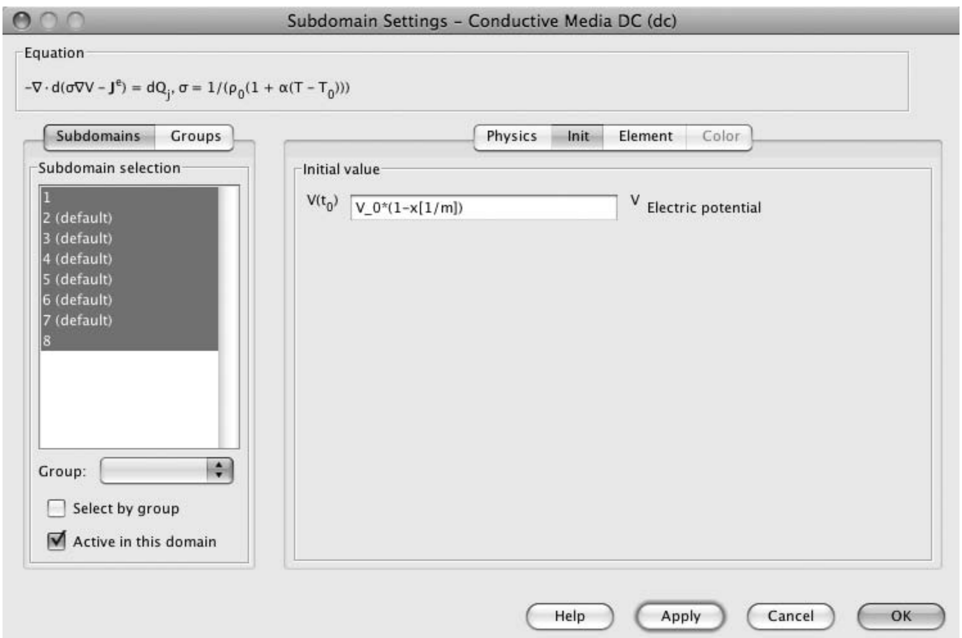


FIGURE 6.50 2D Resistive_Heating_2 model Subdomain Settings, Init edit window

Table 6.13 Boundary Settings–Conductive Media DC (dc) Edit Window

Boundary	Boundary Condition	Value/ Expression	Figure Number
1	Electric potential	V_0	6.51
2, 3, 5–7, 9–11, 13–15, 17–19, 21–23, 25, 26, 28, 29, 31, 33, 35, 37, 39	Electric insulation	—	6.52
40	Ground	—	6.53

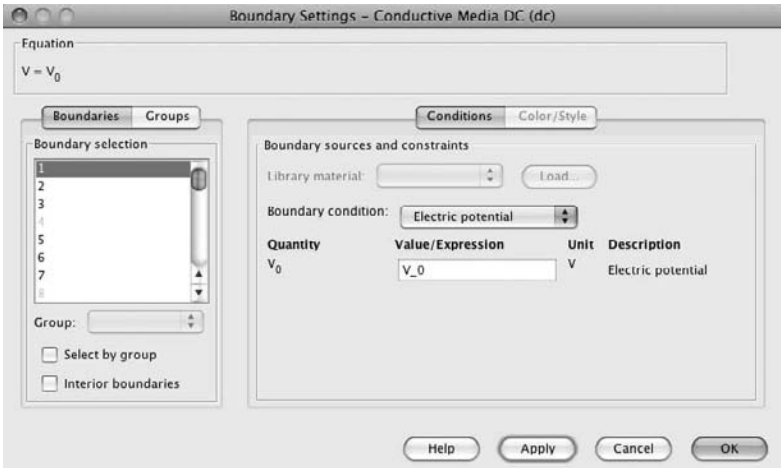


FIGURE 6.51 2D Resistive_Heating_2 model Boundary Settings (1) edit window

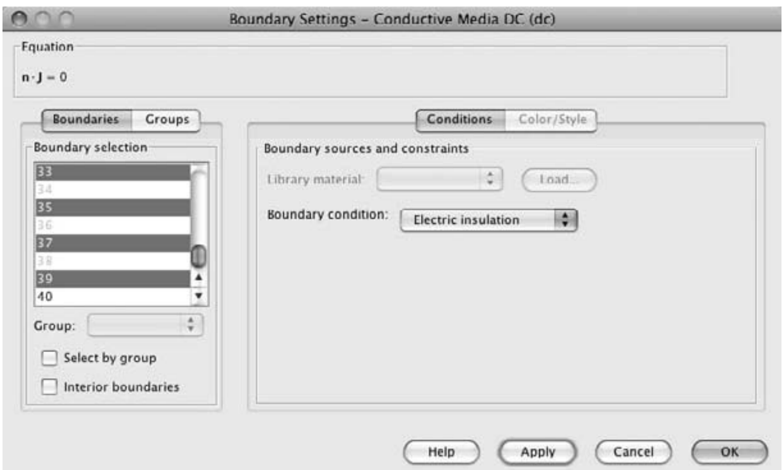


FIGURE 6.52 2D Resistive_Heating_2 model Boundary Settings (2, 3, 5–8…) edit window

FIGURE 6.53 2D Resistive_Heating_2 model Boundary Settings (40) edit window

Click OK. Figure 6.54 shows the 2D Resistive_Heating_2 model with all the boundary settings.

Mesh Generation

On the toolbar, click the Initialize Mesh button once. Click the Refine Mesh button once. This results in a mesh of approximately 2200 elements. See Figure 6.55.

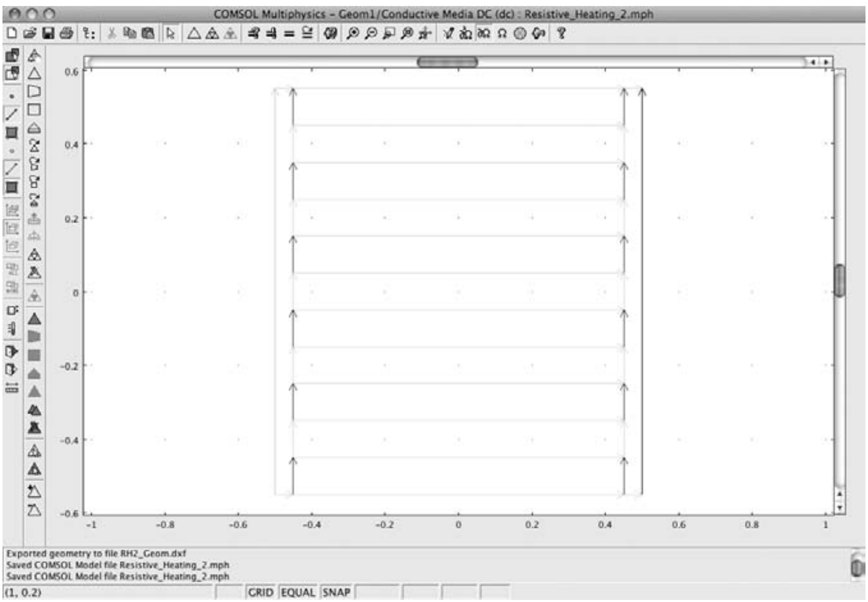


FIGURE 6.54 2D Resistive_Heating_2 model with all the boundary settings

FIGURE 6.55 2D Resistive_Heating_2 model mesh window

Solving the 2D Resistive_Heating_2 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Enter 0:50:2000 in the Times edit window. See Figure 6.56. Click OK.

Time-Dependent Solving of the 2D Resistive_Heating_2 Model

Select Solve > Solve Problem. See Figure 6.57.

Postprocessing and Visualization

The default plot shows the temperature distribution in kelvins. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot

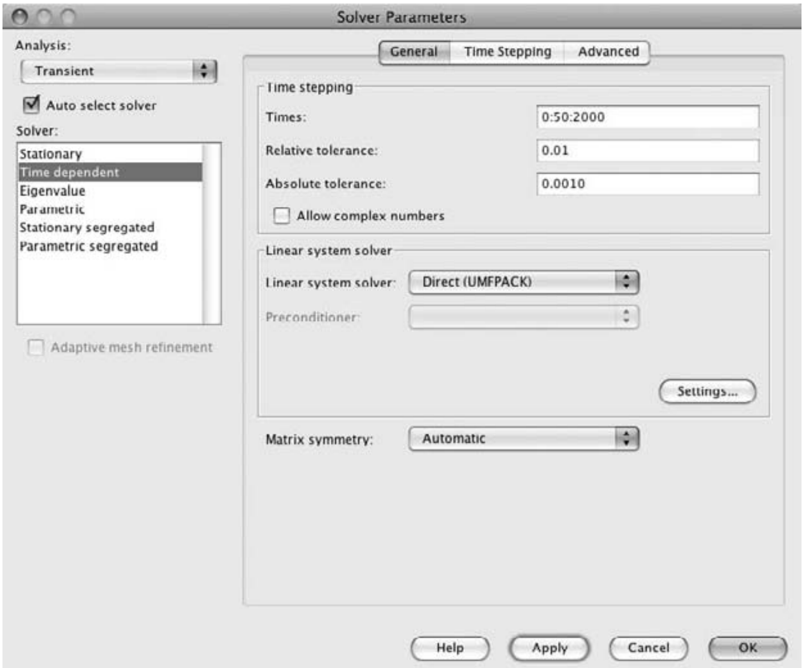


FIGURE 6.56 2D Resistive_Heating_2 model Solver Parameters edit window

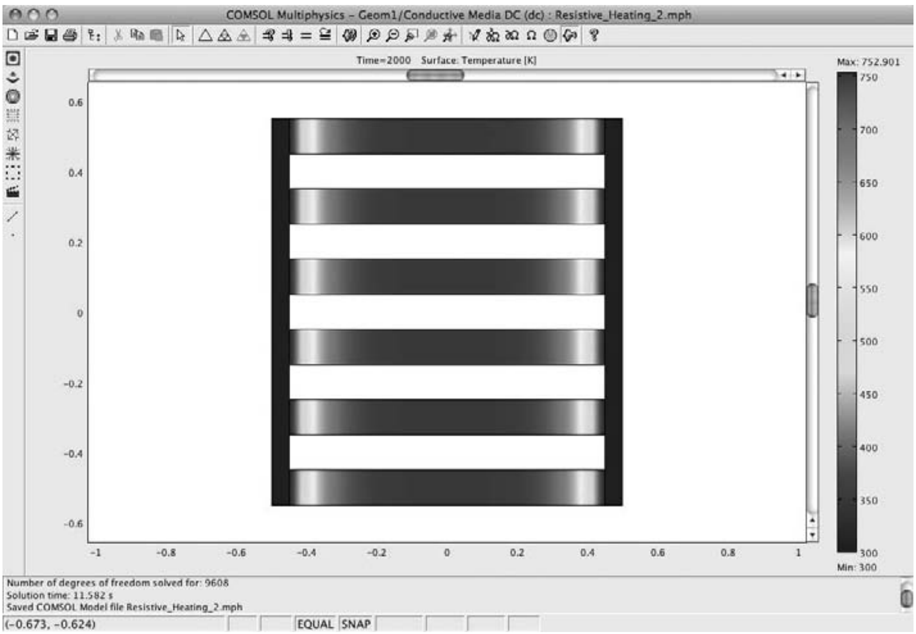


FIGURE 6.57 2D Resistive_Heating_2 model solution

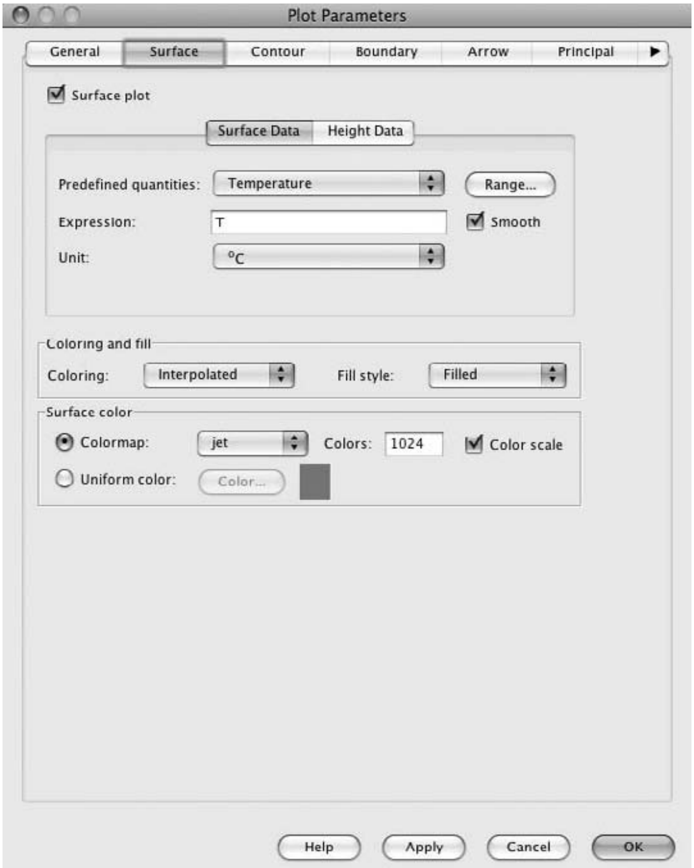


FIGURE 6.58 2D Resistive_Heating_2 model Plot Parameters edit window

Parameters > Surface. Verify that the Surface plot check box is checked and that the Predefined quantities pull-down list shows "Temperature." Select "degC" or "°C" from the Unit pull-down list. See Figure 6.58.

Click OK. See Figure 6.59.

Next, because this is a transient analysis model, the modeler can test how close the solution is to the steady-state value. Select Postprocessing > Cross-Section Plot Parameters > General > Point plot. Verify that all of the Solutions to use are selected. See Figure 6.60.

Click the Point tab. Enter x = 0, y = 0.1 in the Coordinates edit windows. See Figure 6.61.

Click OK. Figure 6.62 shows the temperature versus time plot for the point $x = 0, y = 0.1$. It is easily seen that the temperature is not close to the steady-state value (the curve is still rising, an almost linear $\triangle T$) at the end of the modeling calculation.

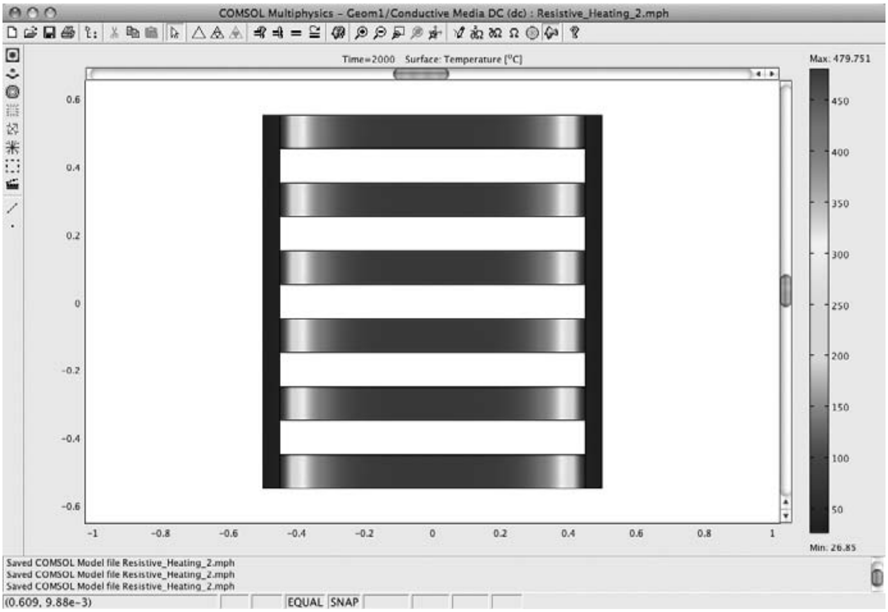


FIGURE 6.59 2D Resistive_Heating_2 model, degrees centigrade

FIGURE 6.60 2D Resistive_Heating_2 model, Cross-Section Plot Parameters, General edit window

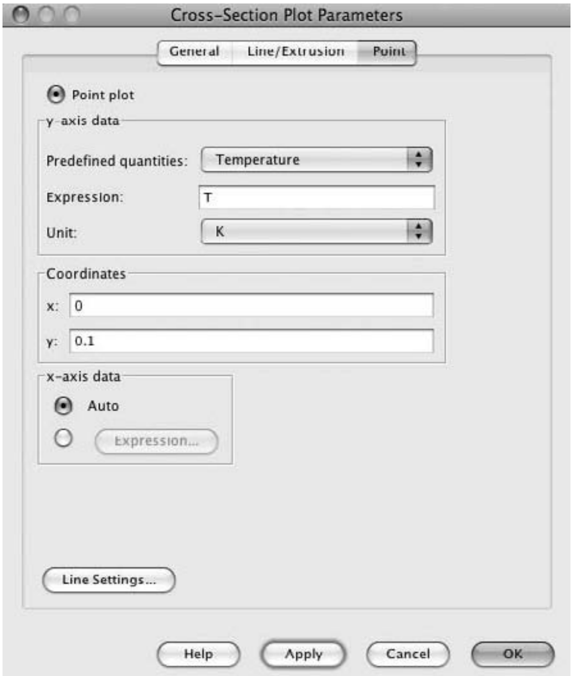


FIGURE 6.61 2D Resistive_Heating_2 model, Cross-Section Plot Parameters, Point edit window

FIGURE 6.62 2D Resistive_Heating_2 model, temperature versus time at $x = 0$, $y = 0.1$

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.63.

Click the Start Animation button. See Figure 6.64.

Alternatively, you can play the file Movie6_RH_2.avi that was supplied with this book.

Second Variation on the 2D Resistive Heating Model, Including Alumina Isolation

The following numerical solution model (Resistive_Heating_3) is derived from the model Resistive_Heating_2. In this model, geometric and materials composition changes are introduced, such as might be used in a general industrial application. This model introduces Alumina as the thermal and electrical insulator and employs the

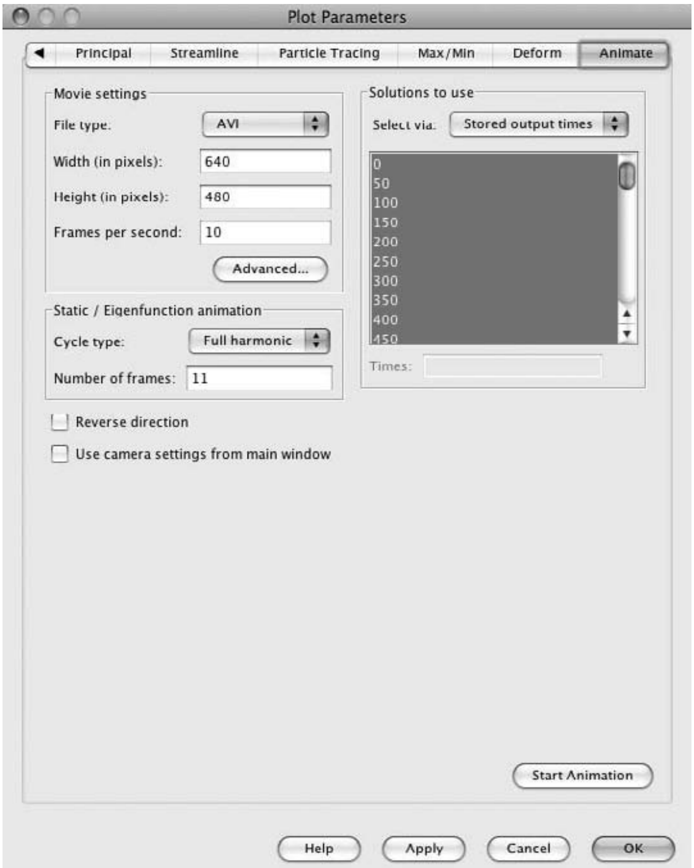


FIGURE 6.63 2D Resistive_Heating_2 model, Plot Parameters window

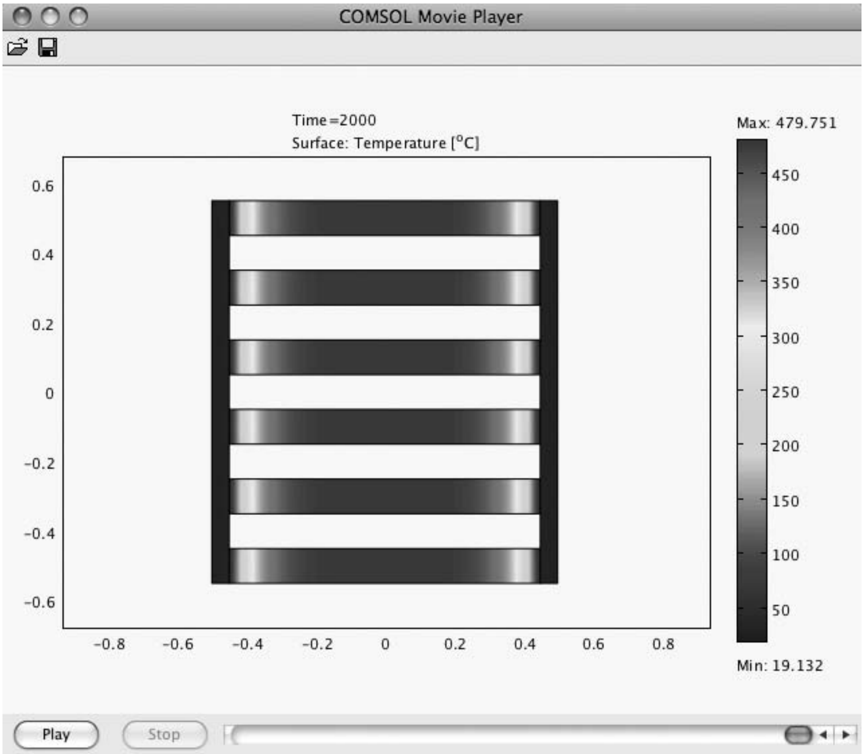


FIGURE 6.64 2D Resistive_Heating_2 model animation, final frame

Heat Transfer Module. It is a multielement heating unit with Nichrome (a nickel–chromium alloy) heating bars and copper connecting bars.

NOTE The Resistive_Heating_3 model demonstrates materials and a configuration as might be employed in vacuum heat sealers, soldering heads, packaging equipment, and other vacuum processing equipment.

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat generation during a process and either add heat or remove heat so as to achieve or maintain a desired temperature. Figure 6.65 shows a 3D rendition of the 2D resistive heating geometry, as will be modeled here.

To start building the Resistive_Heating_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from the Space dimension pull-down list. Select Heat Transfer Module > Electro-Thermal Interaction > Joule Heating > Transient analysis. See Figure 6.66. Click OK.

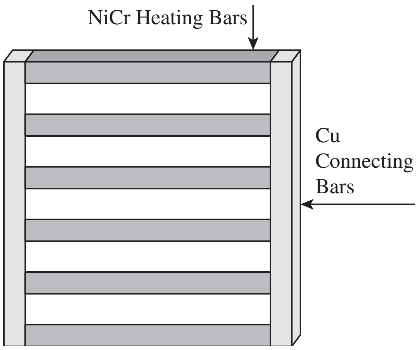


FIGURE 6.65 3D rendition of the 2D Resistive_Heating_3 model (not to scale)

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information as shown in Table 6.14; also see Figure 6.67. Click OK.

NOTE In building this model, the calculational parameters (e.g., constants, scalar expressions) have been consolidated into a convenient location (e.g., a Constants File, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the

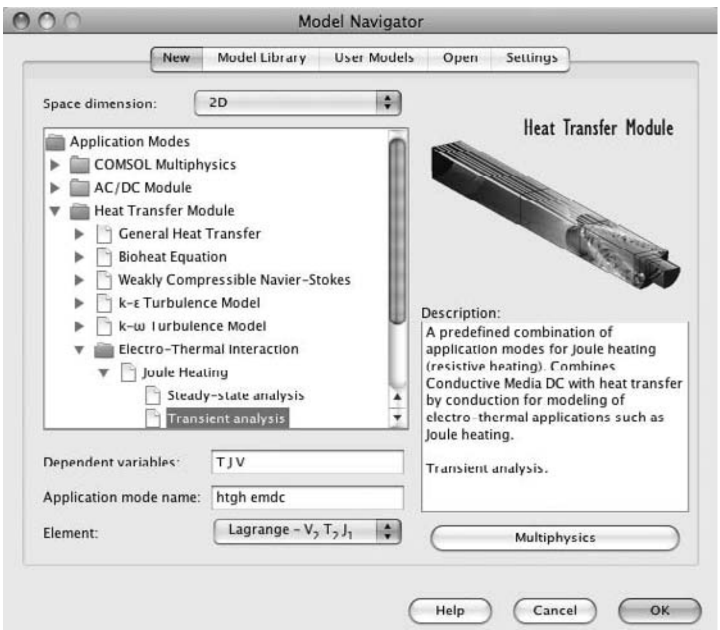


FIGURE 6.66 2D Resistive_Heating_3 Model Navigator setup

Table 6.14 Constants Edit Window

Name	Expression	Description
V_0	1[V]	Electric potential (voltage)
T_ref	20[degC]	Reference temperature
T_air	300[K]	Air temperature
r_Cu	1.754e-8[ohm*m]	Resistivity Cu at T_0
alpha_Cu	3.9e-3[1/K]	Temperature coefficient Cu
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity Cu
rho_Cu	8.96e3[kg/m^3]	Density Cu
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity Cu
r_NiCr	1.08e-6[ohm*m]	Resistivity NiCr at T_0
alpha_NiCr	1.7e-3[1/K]	Temperature coefficient NiCr
k_NiCr	1.13e1[W/(m*K)]	Thermal conductivity NiCr
rho_NiCr	8.4e3[kg/m^3]	Density NiCr
Cp_NiCr	4.5e2[J/(kg*K)]	Heat capacity NiCr

settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

NOTE At this point, three alternate paths can taken. If the geometry was built and exported by building the Resistive_Heating_2 model, then the RH2_Geometry.dxf file can be imported. If not, then the modeler can use the file that comes with the book. However, if the geometry has never been built, then follow the instructions given here.

If one of the import paths is taken, then jump to the next Note.

FIGURE 6.67 2D Resistive_Heating_3 model Constants edit window

Table 6.15 Rectangle Edit Window

R Number	Width	Height	Base	*x*	*y*
1	1.0	1.1	Center	0	0
2	0.9	0.1	Corner	−0.45	0.35
3	0.9	0.1	Corner	−0.45	0.15
4	0.9	0.1	Corner	−0.45	−0.05
5	0.9	0.1	Corner	−0.45	−0.25
6	0.9	0.1	Corner	−0.45	−0.45
7	0.05	1.1	Corner	−0.50	−0.55
8	0.05	1.1	Corner	0.45	−0.55

Using the menu bar, select Draw > Specify Objects > Rectangle. Create each of the rectangles indicated in Table 6.15. See Figure 6.68.

Using the menu bar, select Draw > Create Composite Object. Enter R1+R7+R82−R2−R3−R4−R5−R6. See Figure 6.69.

Click OK, and then click the Zoom Extents button.

FIGURE 6.68 2D Resistive_Heating_3 model created rectangles

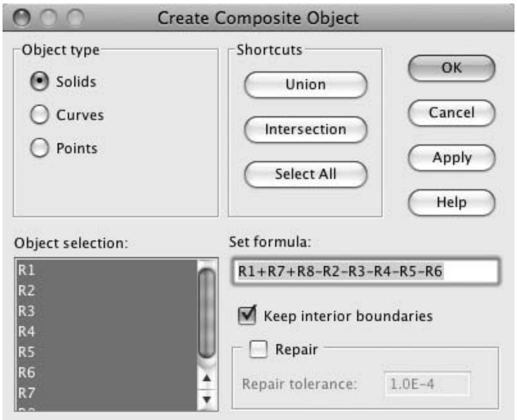


FIGURE 6.69 2D Resistive_Heating_3 model Create Composite Object edit window

NOTE For imported file users, jump to here.

Using the menu bar, select File > Import > CAD Data From File > RH2_Geometry.dxf. Using the menu bar, select Draw > Specify Objects > Rectangle. Create the rectangle indicated in Table 6.16.

Click the Zoom Extents button. See Figure 6.70.

This model introduces the coupling of two basic Application Modes: Joule Heating in the Conductive Media DC Application Mode and the Heat Transfer by Conduction Application Mode. The Physics subdomain and boundary settings will need to be specified in each mode separately. Figure 6.71 shows an overview of the boundary conditions for the combination of both modes.

First, however, the subdomain settings values need to be specified.

Physics Subdomain Settings: General Heat Transfer (htgh)

Having established the geometry for the 2D Resistive_Heating_3 model of a heater bar assembly, the next step is to define the fundamental Physics conditions. Using the menu bar, select Multiphysics > General Heat Transfer (htgh). Using the menu bar, select Physics > Subdomain Settings. Select Subdomains (3, 5, 7, 9, 11) > Load >

Table 6.16 Rectangle Edit Window

R Number	Width	Height	Base	*x*	*y*
1	1.0	1.1	Center	0	0

FIGURE 6.70 2D Resistive_Heating_3 model heater bar assembly

Basic Materials Properties > Alumina. Click OK, and then click the Apply button. See Figure 6.72.

In the Subdomain edit windows, enter the information shown in Table 6.17. Click the Apply button. See Figure 6.73.

In the Subdomain edit windows, enter the information shown in Table 6.18. Click the Apply button. See Figure 6.74.

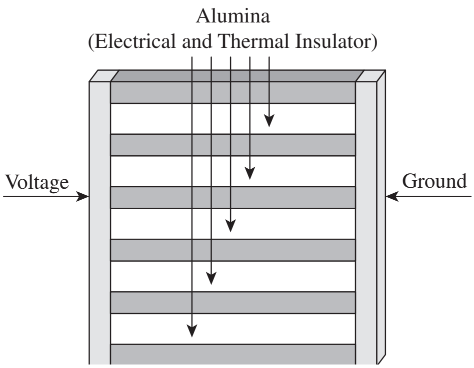


FIGURE 6.71 2D Resistive_Heating_3 model boundary conditions overview (not to scale)

FIGURE 6.72 2D Resistive_Heating_3 model Subdomain Settings (3, 5, 7, 9, 11) edit window

Table 6.17 Subdomain Settings Edit Window

Subdomain Number	Name	Location	Description
1, 13	*k* (isotropic)	k_Cu	Thermal conductivity
	ρ	rho_Cu	Density
	C_P	Cp_Cu	Heat capacity

Table 6.18 Subdomain Settings Edit Window

Subdomain Number	Name	Expression	Description
2, 4, 6, 8, 10, 12	*k* (isotropic)	k_NiCr	Thermal conductivity
	ρ	rho_NiCr	Density
	C_P	Cp_NiCr	Heat capacity

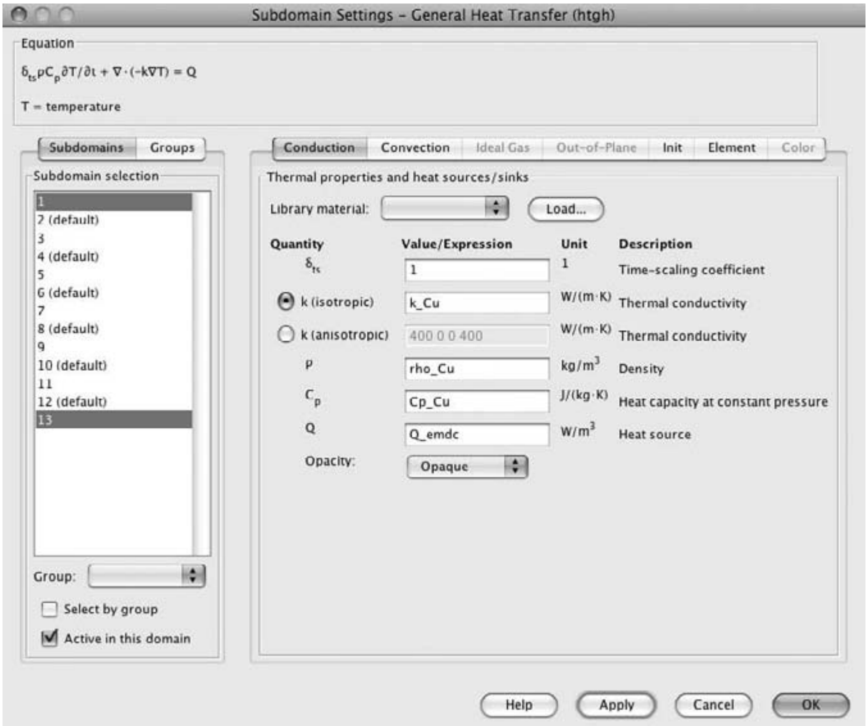


FIGURE 6.73 2D Resistive_Heating_3 model Subdomain Settings (1, 13) edit window

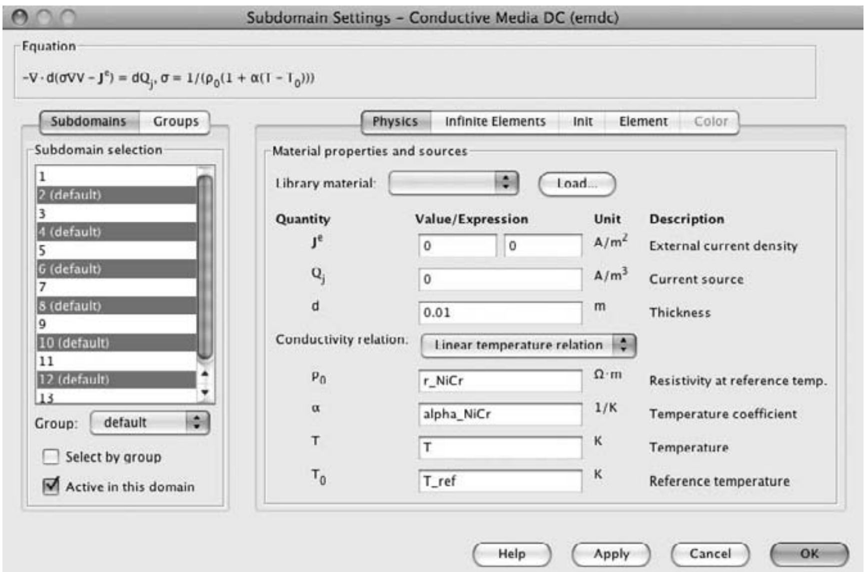


FIGURE 6.74 2D Resistive_Heating_3 model Subdomain Settings (2, 4, 6, 8, 10, 12) edit window

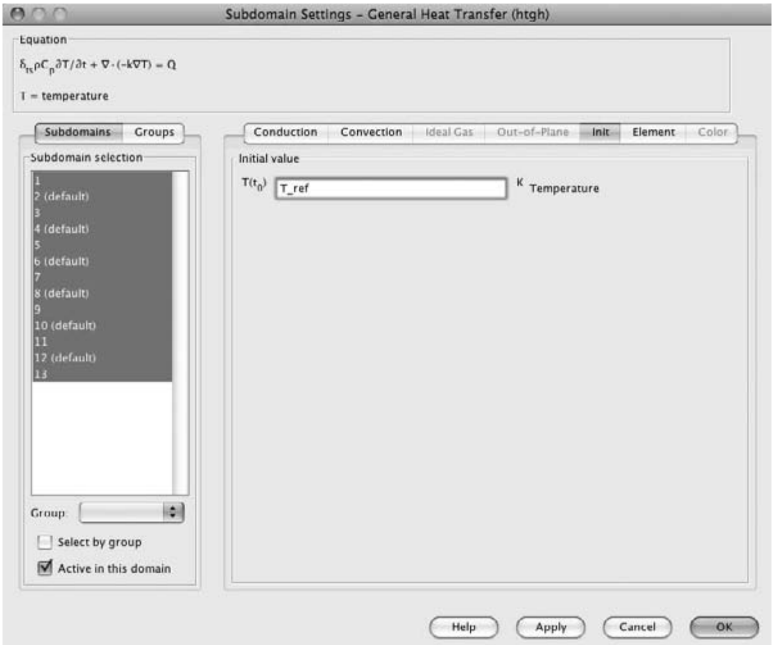


FIGURE 6.75 2D Resistive_Heating_3 model Subdomain Settings, Init edit window

NOTE For transient calculations, all of the physical property values are required for the conduction calculation. In this case, the properties of copper (Cu), Nichrome (NiCr), and Alumina (Al_2O_3) are required.

Select the Init tab. Select subdomains 1–13. Enter T_ref in the Initial value edit window. See Figure 6.75. Click OK.

Physics Boundary Settings: General Heat Transfer (htgh)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 6.19. See Figures 6.76 and 6.77.

Table 6.19 Boundary Settings–Heat Transfer by Conduction (ht) Edit Window

Boundary	Boundary Condition	Value/ Expression	Click Apply	Figure Number
1, 40	Temperature	T_air	Yes	6.76
2, 3, 5, 26, 28, 39	Thermal insulation	—	No	6.77

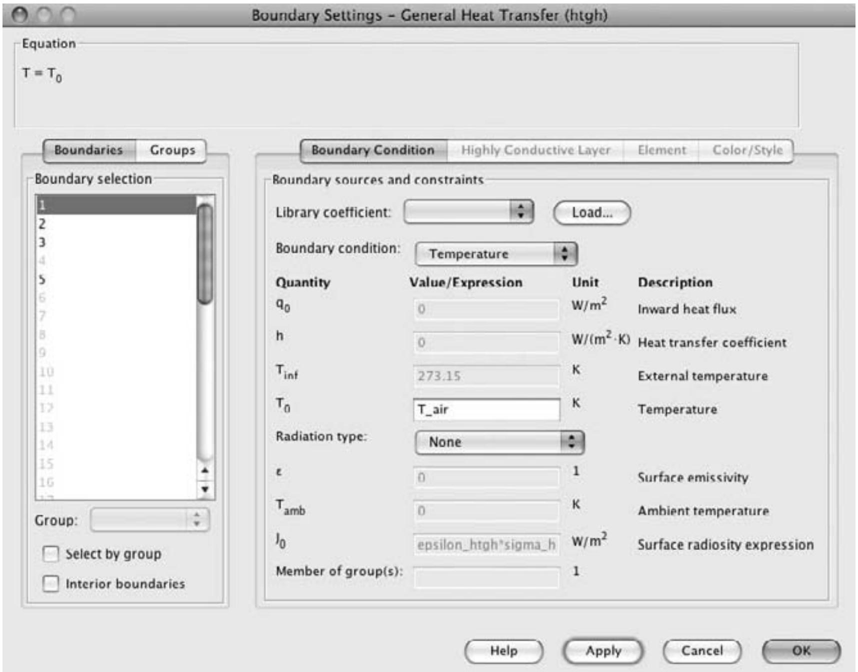


FIGURE 6.76 2D Resistive_Heating_3 model Boundary Settings (1, 40) edit window

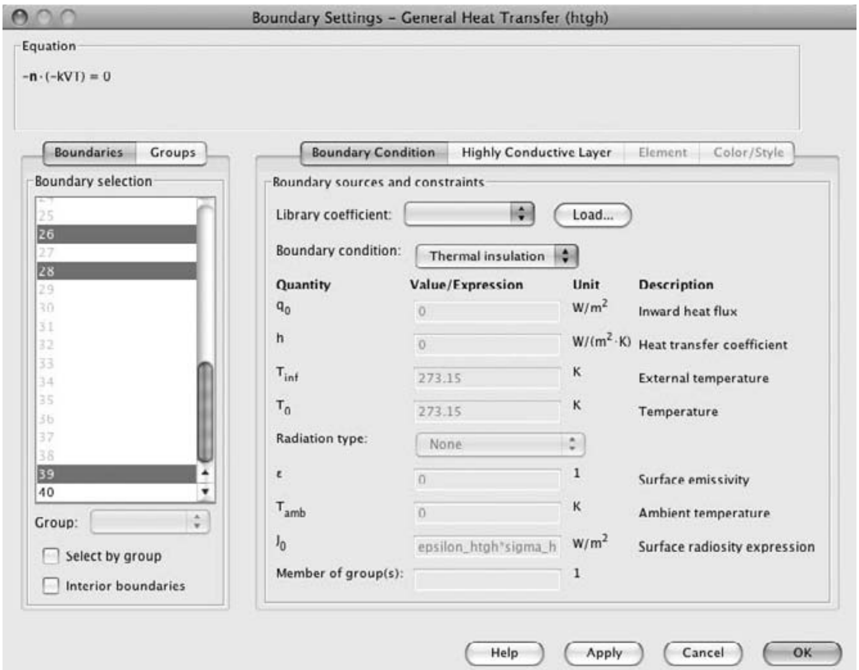


FIGURE 6.77 2D Resistive_Heating_3 model Boundary Settings (2, 3, 5, 26, 28, 39) edit window

Physics Subdomain Settings: Conductive Media DC (emdc)

Using the menu bar, select Multiphysics > Conductive Media DC (dc). Using the menu bar, select Physics > Subdomain Settings. Select subdomains 1–13 in the Subdomain selection window (all of the subdomains). Enter 0.01 in the Thickness (d) edit window. Select “Linear temperature relation” from the Conductivity relation pull-down list. Click the Apply button.

Select Subdomains (3, 5, 7, 9, 11) > Load > Basic Materials Properties > Alumina. Select “Conductivity” from the Conductivity relation pull-down list. Enter 0.001 in the Electric conductivity edit window. Click the Apply button.

In the Subdomain edit windows, enter the information shown in Table 6.20. Click the Apply button. See Figure 6.78.

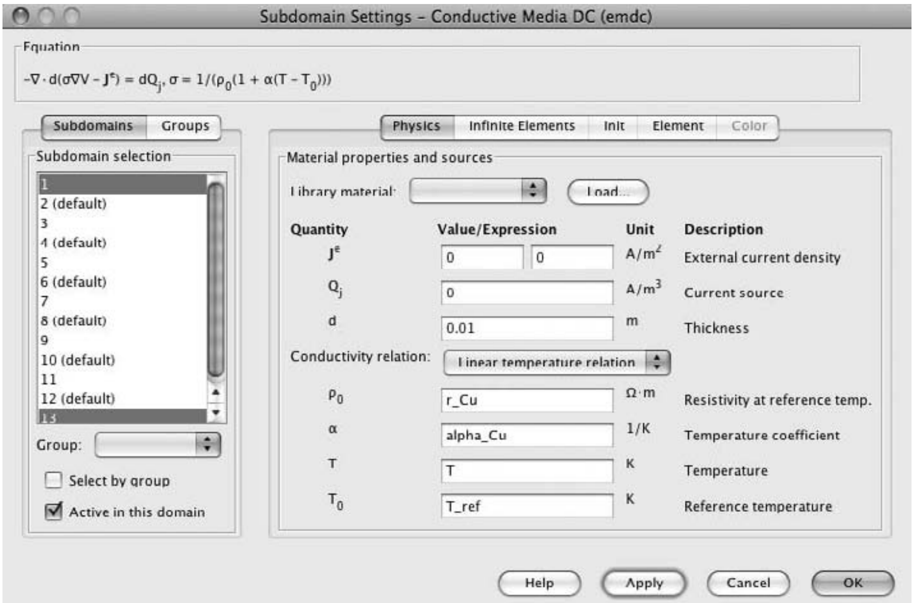


FIGURE 6.78 2D Resistive_Heating_3 model Subdomain Settings (1, 13) edit window

Table 6.20 Subdomain Settings–Conductive Media DC (emdc) Edit Window

Subdomain Number	Name	Expression	Description
1, 13	ρ_0	r_Cu	Resistivity at reference temperature
	α	alpha_Cu	Temperature coefficient
	T_0	T_ref	Reference temperature

Table 6.21 Subdomain Settings–Conductive Media DC (emdc) Edit Window

Subdomain Number	Name	Expression	Description
2, 4, 6, 8, 10, 12	ρ_0	r_NiCr	Resistivity at reference temperature
	α	alpha_NiCr	Temperature coefficient
	T_0	T_ref	Reference temperature

In the Subdomain edit windows, enter the information shown in Table 6.21. Click the Apply button. See Figure 6.79.

NOTE At this point in the model, the generation of heat is coupled to the resistivity in each different material (Cu, NiCr) through the temperature change.

Select the Init tab. Select subdomains 1–13 in the Subdomain selection window (all of the subdomains). Enter V_0*(1 − x[1/m]) in the V(t_0) edit window. See Figure 6.80. Click OK.

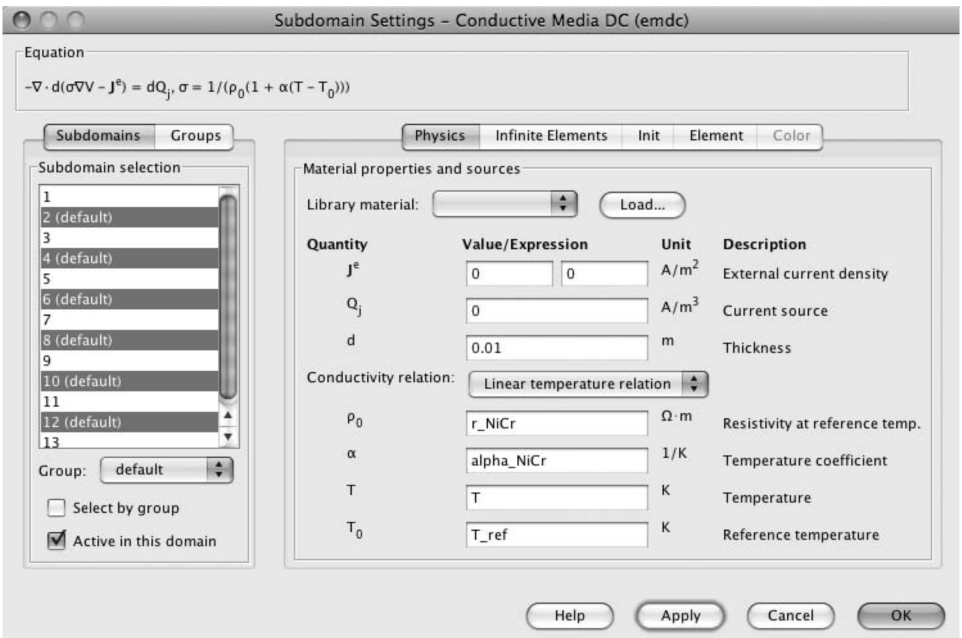


FIGURE 6.79 2D Resistive_Heating_3 model Subdomain Settings (2, 4, 6, 8, 10, 12) edit window

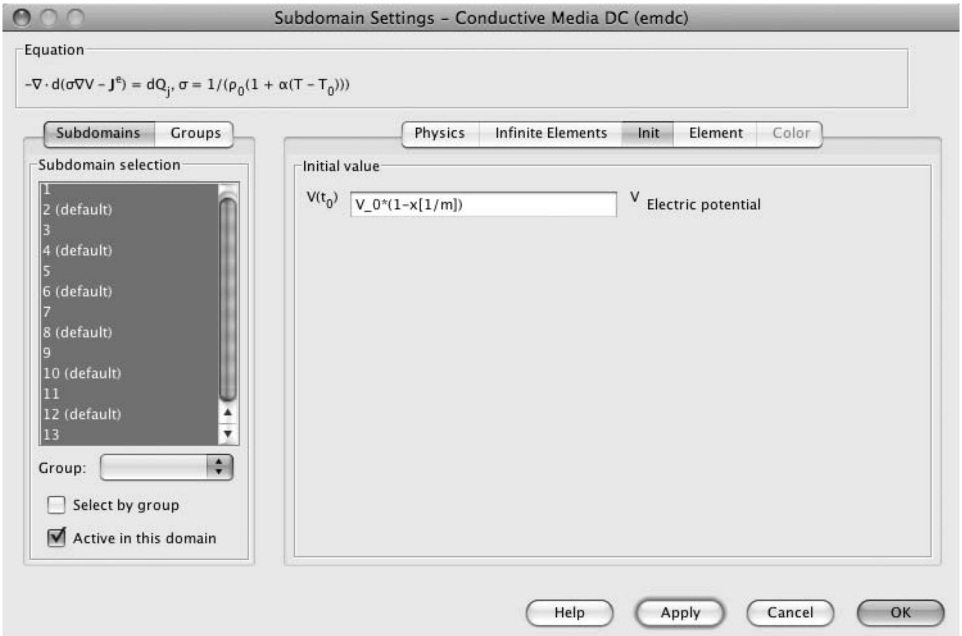


FIGURE 6.80 2D Resistive_Heating_3 model Subdomain Settings Init edit window

NOTE The initial conditions assume a linear voltage drop across the body of the model. This is reflected in the initialization equation ($V(t_0) = V\_0*(1 - x[1/m])$).

Physics Boundary Settings: Conductive Media DC (emdc)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, Select or enter the given boundary condition and value as shown in Table 6.22. Check the Interior boundaries check box. See Figures 6.81, 6.82, and 6.83.

Table 6.22 Boundary Settings–Conductive Media DC (emdc) Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
1	Electric potential	V_0	6.81
2, 3, 5, 26, 28, 39	Electric insulation	—	6.82
40	Ground	—	6.83

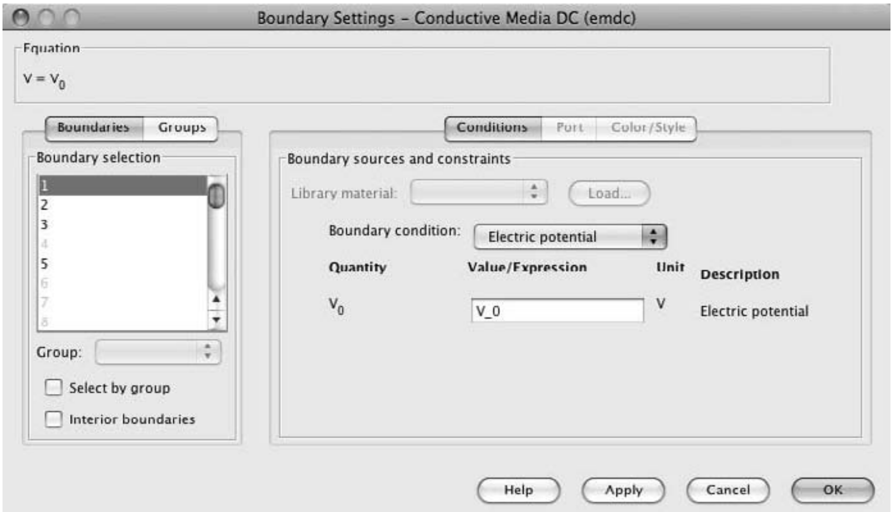


FIGURE 6.81 2D Resistive_Heating_3 model Boundary Settings (1) edit window

Click OK. See Figure 6.84.

Mesh Generation

On the toolbar, click Mesh > Free Mesh Parameters. Select the Subdomain tab. Select subdomains 1–13. Enter 0.02 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 6.85.

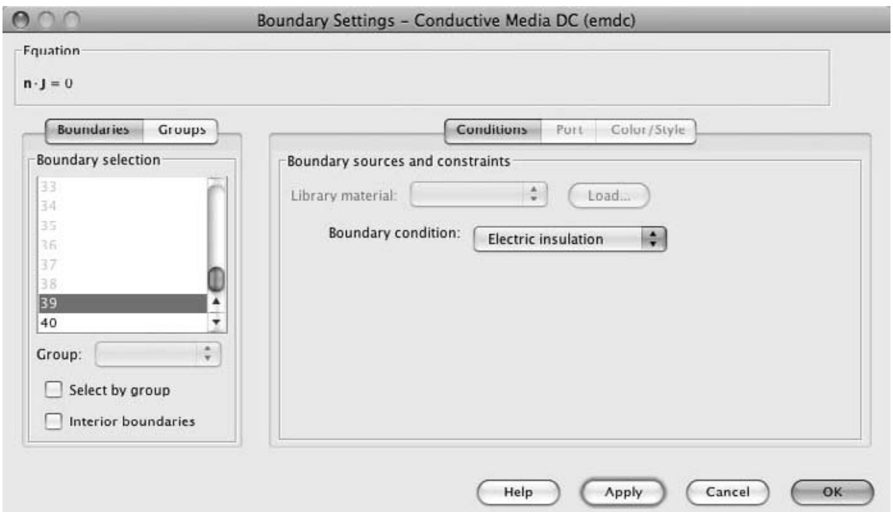


FIGURE 6.82 2D Resistive_Heating_3 model Boundary Settings (2, 3, 5, 26, 28, 39) edit window

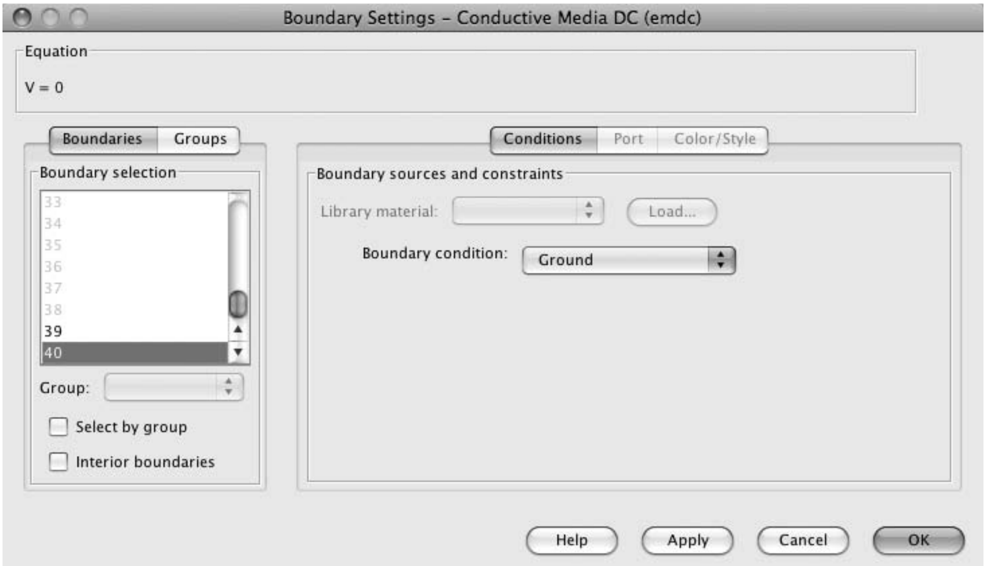


FIGURE 6.83 2D Resistive_Heating_3 model Boundary Settings (40) edit window

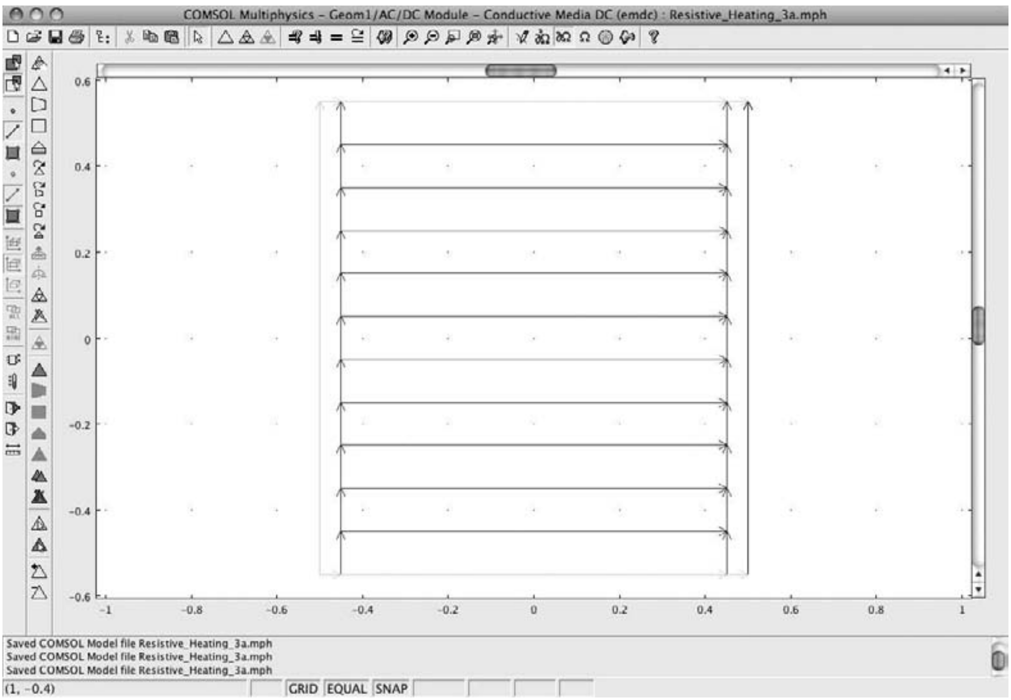


FIGURE 6.84 2D Resistive_Heating_3 model with all the boundary settings

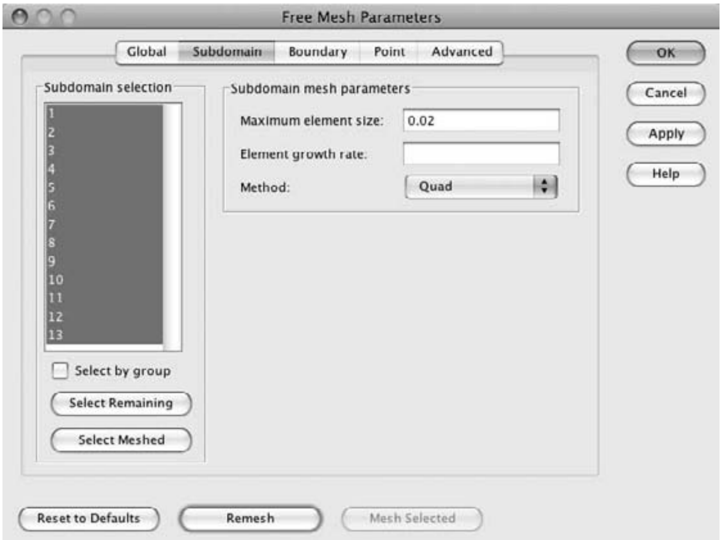


FIGURE 6.85 2D Resistive_Heating_3 model Free Mesh Parameters window

Click the Remesh button. Click OK. See Figure 6.86.

Solving the 2D Resistive_Heating_3 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

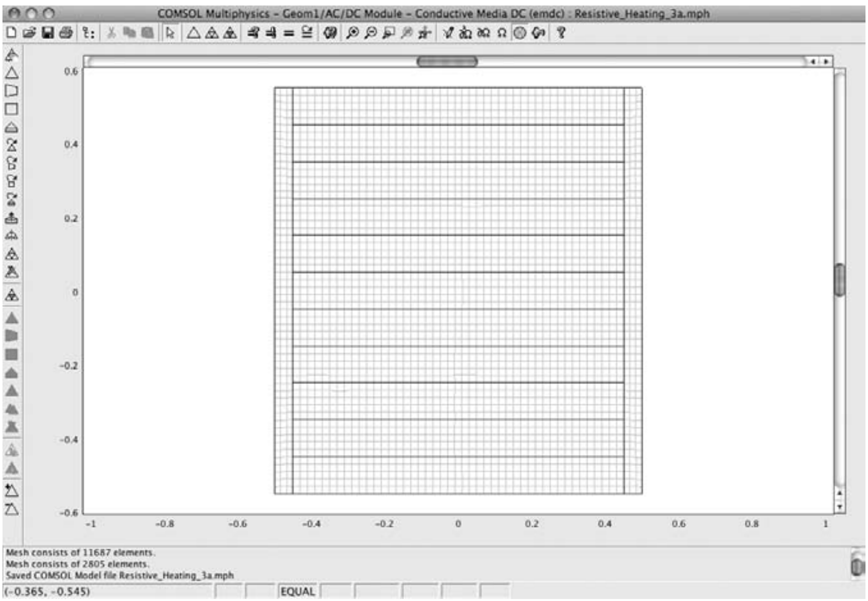


FIGURE 6.86 2D Resistive_Heating_3 model mesh

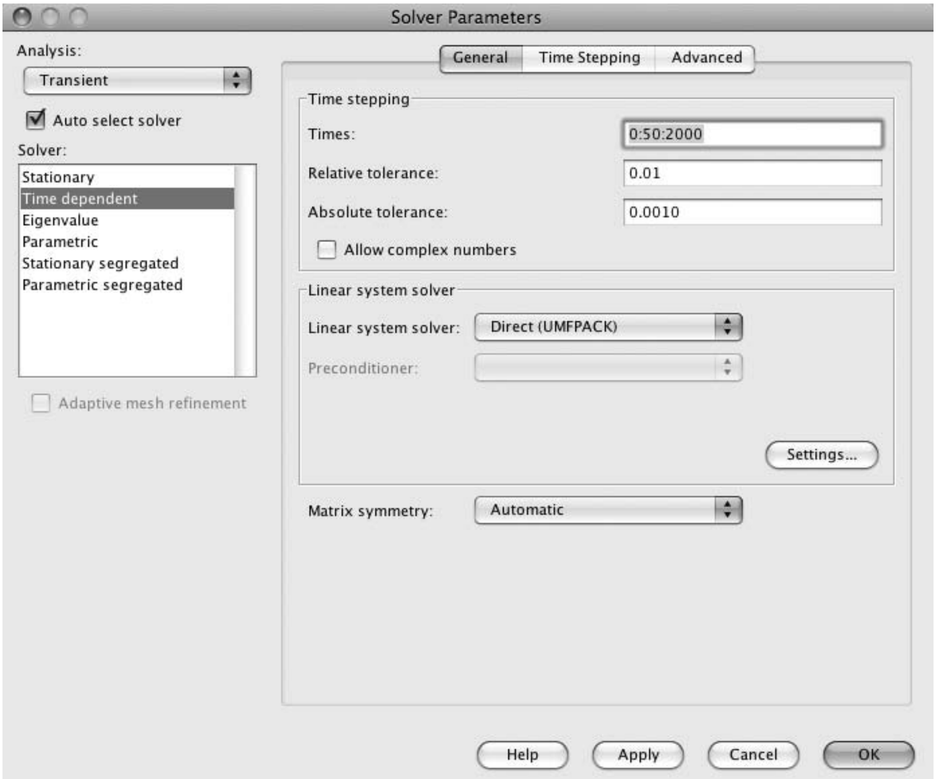


FIGURE 6.87 2D Resistive_Heating_3 model Solver Parameters edit window

---

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Enter 0:50:2000 in the Times edit window. See Figure 6.87. Click OK.

Time-Dependent Solving of the 2D Resistive_Heating_3 Model

Select Solve > Solve Problem. See Figure 6.88.

Postprocessing and Visualization

The default plot shows the temperature distribution in kelvins. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot

FIGURE 6.88 2D Resistive_Heating_3 model solution

Parameters > Surface. Verify that the Surface plot check box is checked and that the Predefined quantities pull-down list shows "Temperature." Select "degC" or "°C" from the Unit pull-down list. See Figure 6.89.

Click OK. See Figure 6.90.

Next, because this is a transient analysis model, the modeler can test how close the solution is to the steady-state value. Select Postprocessing > Cross-Section Plot Parameters > General > Point plot. Verify that all of the Solutions to use are selected. See Figure 6.91.

Click the Point tab. Enter x = 0, y = 0.1 in the Coordinates edit windows. See Figure 6.92.

Click OK. Figure 6.93 shows the temperature versus time plot for the point $x = 0, y = 0.1$. It is easily seen that the temperature is somewhat close to the steady-state value (the curve is still rising, at a decreasing ΔT) at the end of the modeling calculation.

It is interesting to see how the heat flux moves in this array. Select Postprocessing > Plot Parameters > Arrow. Check the Arrow plot check box. Select "Total heat flux."

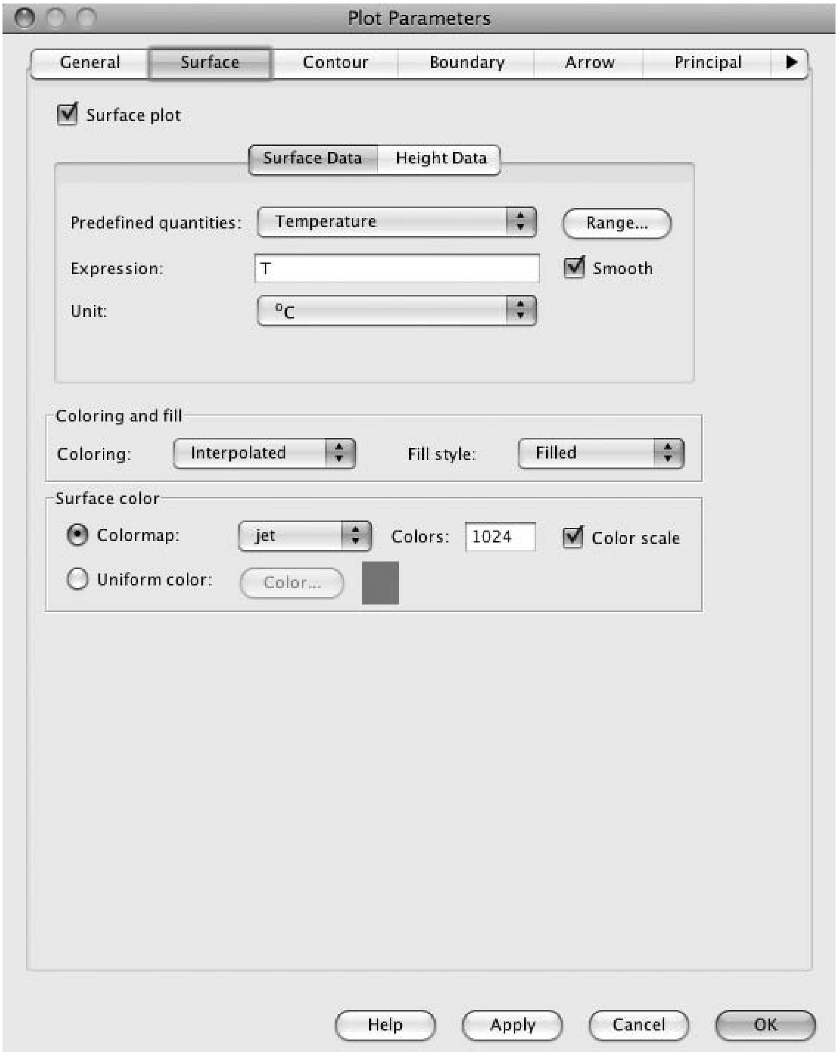


FIGURE 6.89 2D Resistive_Heating_3 model Plot Parameters edit window

Click the Color button and select a color (black). Click OK, and then click OK again. See Figure 6.94.

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.95.

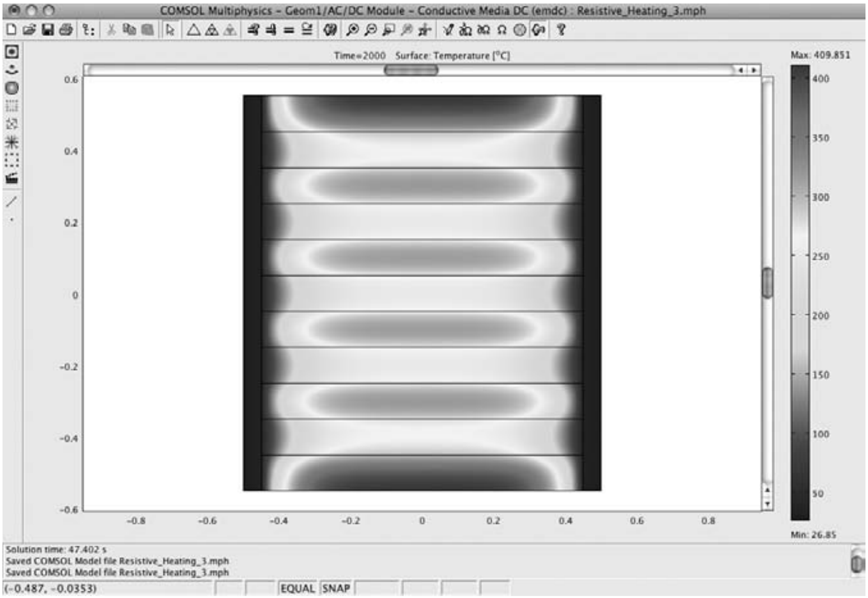


FIGURE 6.90 2D Resistive_Heating_3 model, degrees Centigrade

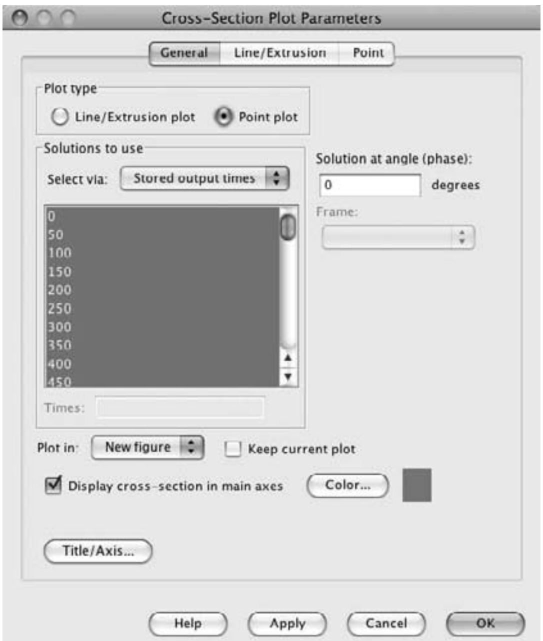


FIGURE 6.91 2D Resistive_Heating_3 model, Cross-Section Plot Parameters, General edit window

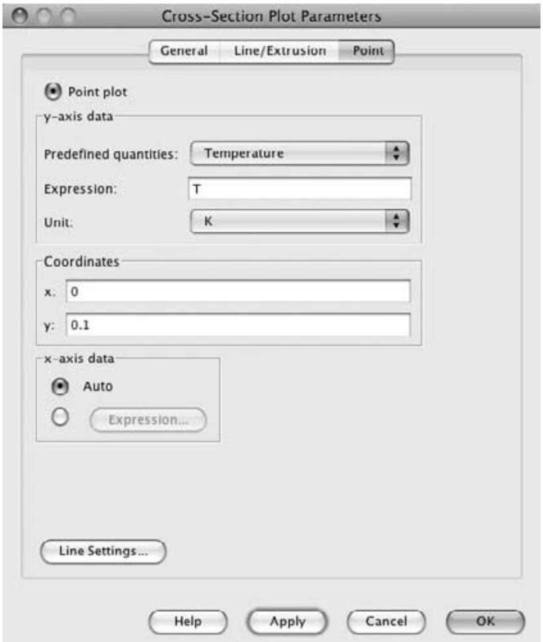


FIGURE 6.92 2D Resistive_Heating_3 model, Cross-Section Plot Parameters, Point edit window

FIGURE 6.93 2D Resistive_Heating_3 model, temperature versus time at $x = 0$, $y = 0.1$

FIGURE 6.94 2D Resistive_Heating_3 model, heat flux (proportional arrows)

Click the Start Animation button. See Figure 6.96.

Alternatively, you can play the file Movie6_RH_3.avi that was supplied with this book.

2D Resistive Heating Models: Summary and Conclusions

The models presented in this section of Chapter 6 have introduced the following new concepts: Ohm's law, Joule heating, mixed-mode modeling, mixed-materials modeling, transient analysis, and the good first approximation. Previously introduced concepts employed include the triangular mesh, free mesh parameters, subdomain mesh, maximum element size, and quadrilateral mesh (quad).

The three resistive heating models are more illustrative of the mixed-mode modeling concept than they are directly amenable to the comparison of calculated values. They present different examples of the diversity of applied scientific and engineering model designs that can be explored using electro-thermal coupling and transient analysis. These models also demonstrate the significant power of relatively simple physical principles, such as Ohm's law and Joule's law.

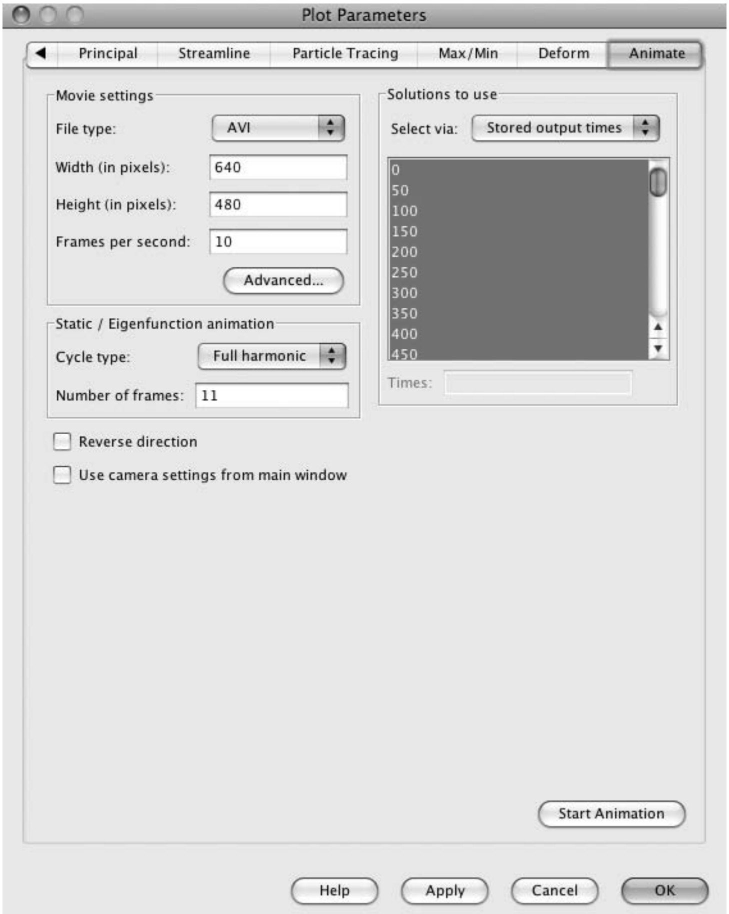


FIGURE 6.95 2D Resistive_Heating_3 model, Plot Parameters window

2D Inductive Heating Considerations

2D Axisymmetric Coordinate System

In this part of Chapter 6, the concepts of interest are most easily explored using the 2D axisymmetric coordinate system. Reviewing briefly the 2D axisymmetric coordinate system basics, parameters in steady-state solutions to any 2D axisymmetric model can vary only as a function of the radial position in space (r) and the axial position in space (z) coordinates. Such models represent the parametric condition of the model in a time-independent mode (quasi-static). In transient solution models, as

FIGURE 6.96 2D Resistive_Heating_3 model animation, final frame

presented in this section, parameters can vary both by position in space (r) and space (z) and by time (t); see Figure 6.97.

The transient solution model is essentially a sequential collection of steady-state (quasi-static) solutions, except that the condition has been added that at least one of the dependent variables changes as a function of time.

NOTE In transient or time-dependent (e.g., dynamic, unsteady) models, at least one of the dependent variables changes as a function of time. The change in the dependent variable is a direct result of the coupling inherent in the physical properties of the materials involved in the model.

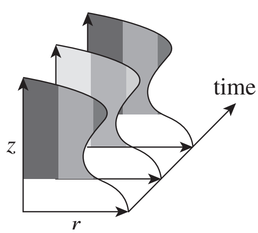


FIGURE 6.97 2D axisymmetric coordinate system, plus time

For example, the resistance of a material typically changes (±) as a function of temperature. When heat is generated in a material through current flow (Joule's law), the temperature of the material changes. Hence, as the temperature changes over time, the resistance changes.

The space coordinates (r) and (z) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values ($r_{\text{min}} <= r <= r_{\text{max}}$) and ($z_{\text{min}} <= z <= z_{\text{max}}$). The time coordinate ($t$) represents the range of values ($t_{\text{min}} <= t <= t_{\text{max}}$) from the beginning of the observation period (t_{min}) to the end of the observation period (t_{max}).

Joule heating techniques are extremely important in device design considerations. Joule heating is applied to tasks as varied as heating houses (AC) and baking potatoes (microwave AC). It accounts for some of the most widely utilized technologies employed for research, design, and application in engineering and physics. Most modern products or processes require an understanding of Joule heating techniques either during development or during the use of the product or process (e.g., automobiles, plate glass fabrication, plastic extrusion, plastic products, houses, baked potatoes, ice cream).

NOTE Heating and heat transfer concerns have existed since the beginning of prehistory. There have been many contributors to the present understanding of the interaction of electric currents and solids. Three scientists especially stand out in this regard: Georg Ohm, James Prescott Joule, and Leon Foucault.

Ohm published his discovery of Ohm's law in 1827:[14]

$$I = \frac{V}{R} \tag{6.5}$$

where I = current in amperes (A)
V = voltage (electromotive force) in volts (V)
R = resistance in ohms

Joule discovered Joule's law in 1843:[15,16]

$$Q = I^2 \cdot R \cdot t \tag{6.6}$$

where Q = heat generated in joules (J)
I = current in amperes (A)
R = resistance in ohms
t = time in seconds (S)

In 1851, Foucault[17] discovered eddy currents[18] (also called Foucault currents). Eddy currents result when a conductor is in the presence of a changing magnetic field.

$$(j\omega\sigma - \omega^2\varepsilon)A + \nabla \times (\mu^{-1}\nabla \times A) = J^{\mathrm{e}} \tag{6.7}$$

where
A = magnetic vector potential
ω = angular frequency
σ = conductivity
ϵ = permittivity
μ = permeability
J^{e} = current density due to an external source[19]

The induced eddy currents interact with the resistance of the conductor (Ohm's law) through Joule's law, causing heat. The net effect of these interactions is induction heating.[20] In this model, the equations are as follows:

$$j\omega\sigma(T)A + \nabla \times (\mu^{-1}\nabla \times A) = 0 \tag{6.8}$$

where
A = magnetic vector potential
ω = angular frequency
$\sigma(T)$ = conductivity
μ = permeability

and

$$\rho C_{\mathrm{p}}\frac{\partial T}{\partial t} - \nabla \cdot k\nabla T = Q(T, A) \tag{6.9}$$

where
ρ = density
C_{p} = specific heat capacity
T = temperature
t = time
k = thermal conductivity
A = magnetic vector potential

and

$$\sigma(T) = (\rho_{\mathrm{ref}}(1 + \alpha(T - T_{\mathrm{ref}}))^{-1} \tag{6.10}$$

where
$\sigma(T)$ = electrical conductivity
ρ_{ref} = resistivity at the reference temperature
T = temperature
T_{ref} = reference temperature
α = thermal coefficient of the resistivity

and

$$Q(T) = \frac{1}{2}\sigma(T)|E_{\mathrm{p}}|^2 \qquad (6.11)$$

where $Q(T)$ = heat generated per period for a sinusoidal wave function
$\sigma(T)$ = conductivity at the present temperature
T = temperature
E_{p} = electric field, peak value

The first example presented in this section, the Inductive_Heating_1 model, explores 2D axisymmetric electro-thermal interaction modeling of Joule heating using transient analysis. The model is solved for a material that is both electrically and thermally conductive. The model is implemented using the COMSOL AC/DC Module Electro-Thermal Application Mode. This model demonstrates the principle of induction heating.

In the first variation on the inductive heating model, the new model is built to explore a common configurational change and is solved using the same COMSOL Multiphysics AC/DC Module Application Mode. In the second variation on this model, a model is built that incorporates materials modifications in addition to the configurational changes and is again solved using the COMSOL Multiphysics AC/DC Electro-Thermal Application Mode. The second variation also explores the influence of an insulating environment on the model's properties. The calculated modeling results are then compared.

2D Axisymmetric Inductive Heating Model

The following numerical solution model (Inductive_Heating_1) is derived from a model that was originally developed by COMSOL as a Multiphysics demonstration model for distribution with the Multiphysics software in the basic Multiphysics Model Library. This model continues the introduction of the coupling of two important basic physical materials properties: Joule heating and heat transfer. The coupling of these two properties in this model demonstrates one of the interactions normally found in typical engineering materials.

NOTE The new modeler should personally build each model. There is no better method to obtain a rapid understanding of the modeling process than to employ the process of gaining the hands-on experience of actually building, meshing, solving, and postprocessing a model. Many times the inexperienced modeler will make and correct errors, thereby adding to his or her experience and fund of modeling knowledge. Even building the simplest model will expand the modeler's store of knowledge.

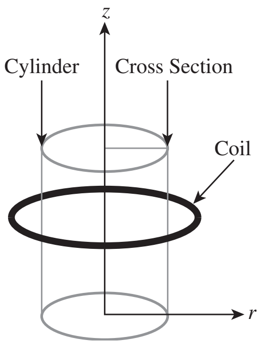


FIGURE 6.98 3D rendition of the 2D axisymmetric Inductive_Heating_1 model

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat generation during a process and either adds heat or removes heat so as to achieve or maintain a desired temperature. Figure 6.98 shows a 3D rendition of the 2D inductive heating geometry, as will be modeled here.

To start building the Inductive_Heating_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select AC/DC Module > Electro-Thermal Interaction > Azimuthal Induction Heating > Transient analysis. See Figure 6.99. Click OK.

FIGURE 6.99 2D axisymmetric Inductive_Heating_1 Model Navigator setup

Table 6.23 Axis Edit Window

Name	Value
r min	−0.05
r max	0.5
z min	−0.3
z max	0.3

Table 6.24 Grid Edit Windows

Name	Value
r spacing	0.05
Extra r	0.03
z spacing	0.05
Extra z	−0.01 0.01

NOTE There are two mode names in the Application mode name window (htgh emqa). The names in the window show which applications are coupled in this application: General Heat Transfer (htgh) and Azimuthal Induction Currents, Vector Potential (emqa).

Options and Settings

Using the menu bar, select Options > Axes/Grid Settings. In the Axis edit windows, enter the information shown in Table 6.23; see Figure 6.100.

Select the Grid tab. Uncheck the Auto check box. In the Grid edit windows, enter the information shown in Table 6.24; See Figure 6.101. Click OK.

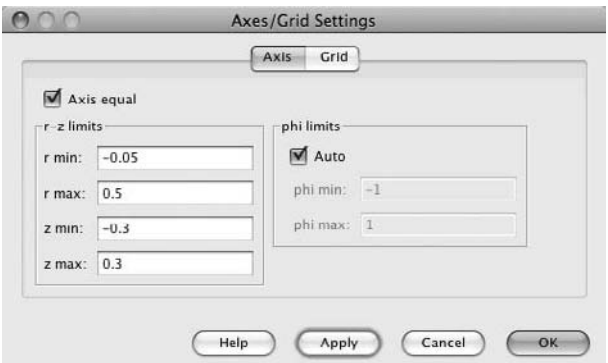


FIGURE 6.100 2D axisymmetric Inductive_Heating_1 model Axis edit window

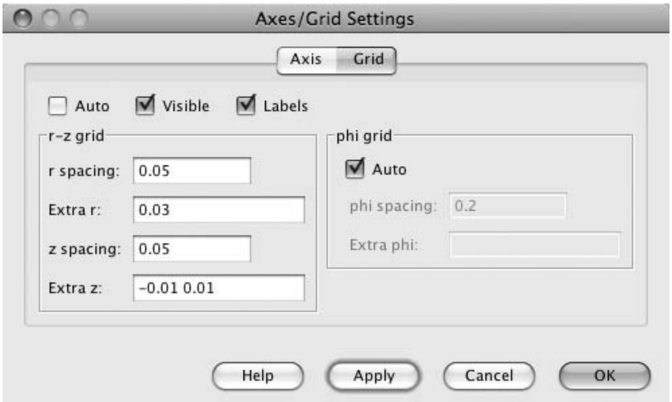


FIGURE 6.101 2D axisymmetric Inductive_Heating_1 model Grid edit window

NOTE The "extra" *r* and *z* grid points are defined points on the grid that are used to aid the modeler in creating the designs needed to build this model. The model can also be built on a standard grid by using the Specify Object commands.

NOTE When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 6.25; also see Figure 6.102. Click OK.

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.01. Select "Base: Center" and set r equal to 0.05 and z equal to 0 in the Circle edit window. See Figure 6.103. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.2 and a height of 0.5. Select "Base: Corner" and set r equal to 0 and z equal to −0.25 in the Rectangle edit window. See Figure 6.104. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.03 and a height of 0.1. Select "Base: Corner" and set r equal to 0 and z equal to −0.05 in the Rectangle edit window. See Figure 6.105.

Table 6.25 Constants Edit Window

Name	Expression	Description
I_0	1e3[A]	Coil current
d_0	2e-2[m]	Coil diameter
c_0	pi*d_0	Coil circumference
Js_0	I_0/c_0	Coil surface current density
T_ref	20[degC]	Reference temperature
r_Cu	1.67e-8[ohm*m]	Resistivity copper
alpha_Cu	3.9e-3[1/K]	Resistivity coefficient copper
rho_air	1.293[kg/m^3]	Density air STP
Cp_air	1.01e3[J/(kg*K)]	Heat capacity air
k_air	2.6e-2[W/(m*K)]	Thermal conductivity air
rho_Cu	8.96e3[kg/m^3]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity copper
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity copper

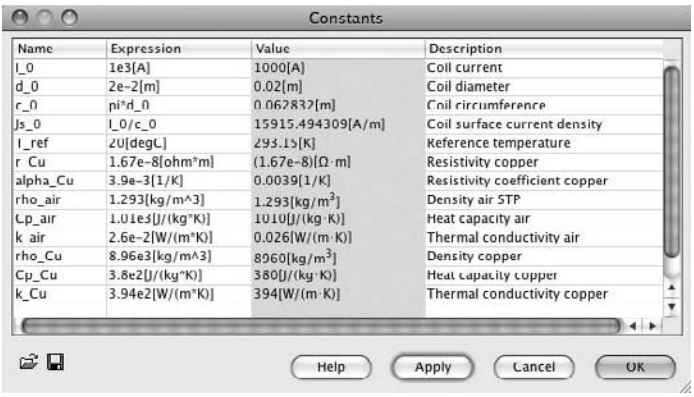


FIGURE 6.102 2D axisymmetric Inductive_Heating_1 model Constants edit window

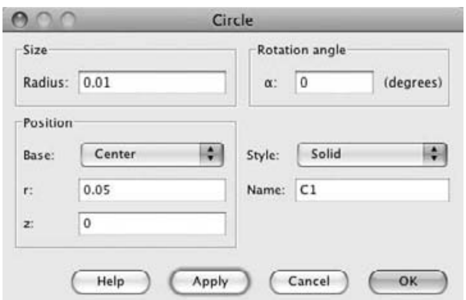


FIGURE 6.103 2D axisymmetric Inductive_Heating_1 model Circle edit window

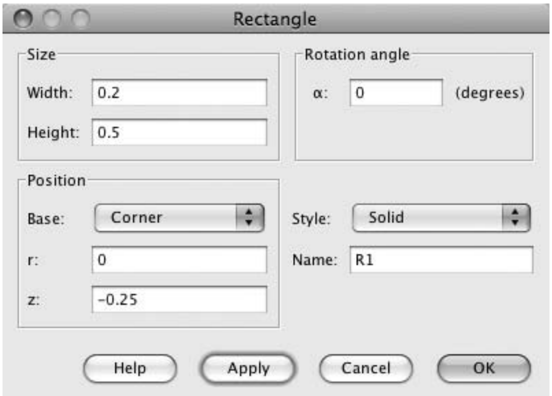


FIGURE 6.104 2D axisymmetric Inductive_Heating_1 model Rectangle edit window

Click OK, and then click the Zoom Extents button. See Figure 6.106.

NOTE The rectangle (R2) is the 2D representation of a cylinder in cross section. The circle is the cross-section profile of the current loop (coil), as was shown earlier in Figure 6.98.

Physics Settings

Select Physics > Scalar Variables. Enter 500 in the nu_emqa edit window. See Figure 6.107. Click OK.

Select Options > Expressions > Scalar Expressions. Enter the scalar expression for sigma_T as shown in Table 6.26; also see Figure 6.108. Click OK.

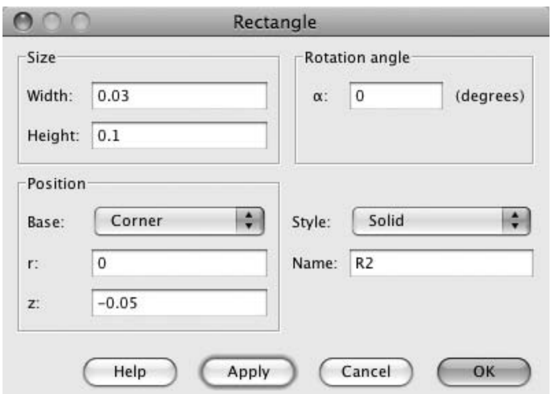


FIGURE 6.105 2D axisymmetric Inductive_Heating_1 model Rectangle edit window

FIGURE 6.106 2D axisymmetric Inductive_Heating_1 model

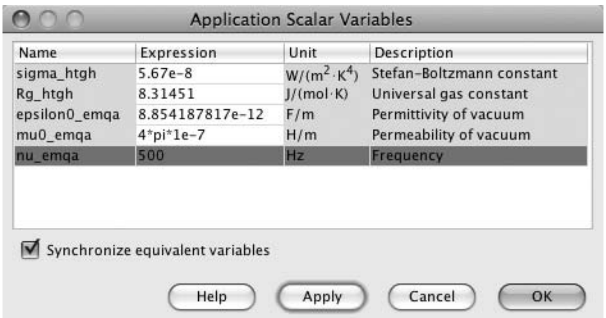


FIGURE 6.107 2D axisymmetric Inductive_Heating_1 model Application Scalar Variables edit window

Table 6.26 Scalar Expressions Edit Window

Name	Expression	Description
sigma_T	1/(r_Cu*(1 + alpha_Cu*(T-T_ref)))	Electrical conductivity copper

FIGURE 6.108 2D axisymmetric Inductive_Heating_1 model Scalar Expressions edit window

NOTE The scalar expression for sigma_T couples the resistivity of copper (r_Cu) and the temperature (T).

Physics Subdomain Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Multiphysics > Azimuthal Induction Currents, Vector Potential (emqa). Using the menu bar, select Physics > Subdomain Settings > Electric Parameters. In the Subdomain edit windows, enter the information shown in Table 6.27; also see Figure 6.109. Click OK.

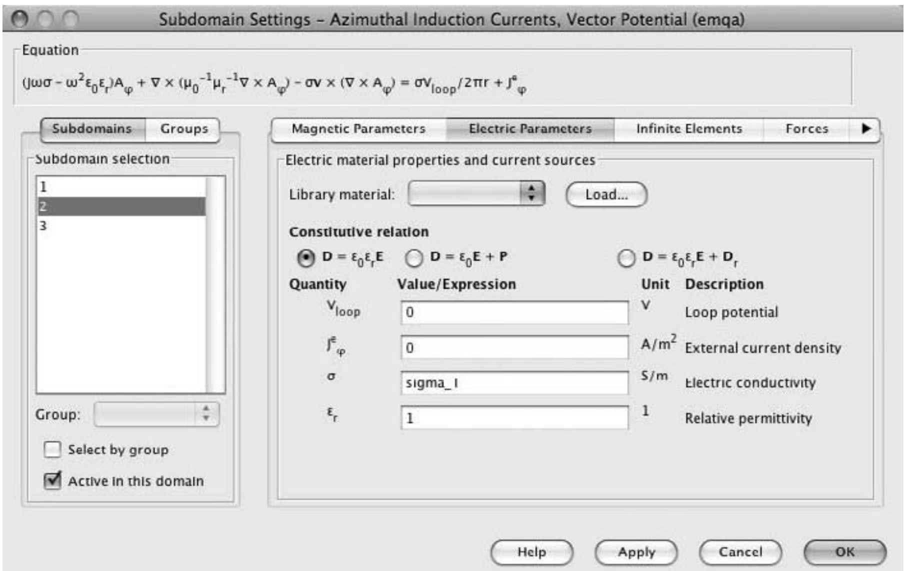


FIGURE 6.109 2D axisymmetric Inductive_Heating_1 model Subdomain Settings (2), Electric Parameters edit window

Table 6.27 Subdomain Edit Window

Subdomain	Name	Expression	Description
1	σ	0	Electric conductivity
2	σ	sigma_T	Electric conductivity
3	σ	0	Electric conductivity

Table 6.28 Boundary Settings Edit Window

Boundary	Boundary Condition	Figure Number
1, 3, 5	Axial symmetry	6.110
2, 7, 9	Magnetic insulation	6.111

Physics Boundary Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.28. Check the Interior boundaries check box. See Figures 6.110 and 6.111.

FIGURE 6.110 2D axisymmetric Inductive_Heating_1 model Boundary Settings (1, 3, 5) edit window

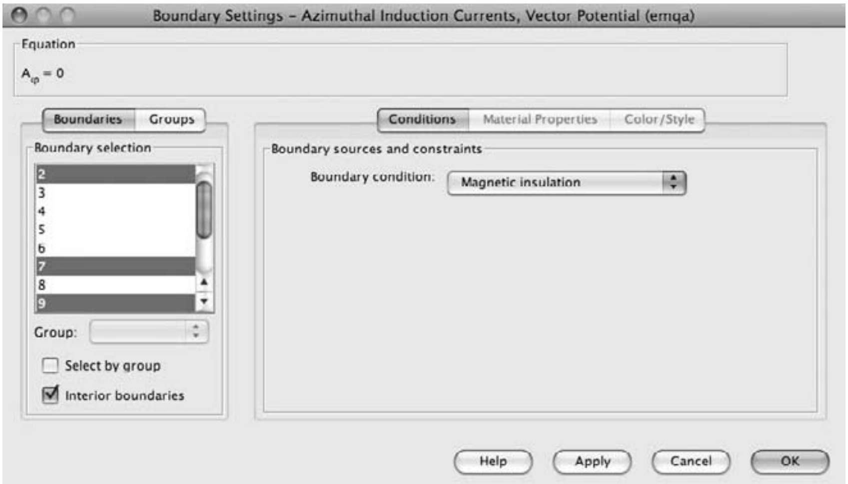


FIGURE 6.111 2D axisymmetric Inductive_Heating_1 model Boundary Settings (2, 7, 9) edit window

In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.29. Click OK. See Figure 6.112.

Table 6.29 Boundary Settings Edit Window

Boundary	Boundary Condition	Value	Figure Number
10–13	Surface current	Js_0	6.112

FIGURE 6.112 2D axisymmetric Inductive_Heating_1 model Boundary Settings (10–13) edit window

Table 6.30 Subdomain Edit Window

Subdomain	Name	Expression	Description	Figure Number
1	k (isotropic)	k_air	Thermal conductivity	6.113
	ρ	rho_air	Density	
	C_p	Cp_air	Heat capacity	
2, 3	k (isotropic)	k_Cu	Thermal conductivity	6.114
	ρ	rho_Cu	Density	
	C_p	Cp_Cu	Heat capacity	

Physics Subdomain Settings: General Heat Transfer (htgh)

Using the menu bar, select Multiphysics > General Heat Transfer (htgh). Using the menu bar, select Physics > Subdomain Settings > Conduction. In the Subdomain edit windows, enter the information shown in Table 6.30. See Figures 6.113 and 6.114.

FIGURE 6.113 2D axisymmetric Inductive_Heating_1 model Subdomain Settings (1) edit window

FIGURE 6.114 2D axisymmetric Inductive_Heating_1 model Subdomain Settings (2, 3) edit window

Select the Init tab. Select all subdomains (1–3). Enter T_ref in the $T(t_0)$ edit window. See Figure 6.115. Click OK.

Physics Boundary Settings: General Heat Transfer (htgh)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – General Heat Transfer (htgh) edit windows, enter the information shown in Table 6.31. Click OK. See Figures 6.116 and 6.117.

Table 6.31 Boundary Settings Edit Window

Boundary	Boundary Condition	Value	Figure Number
1, 3, 5	Axial symmetry	—	6.116
2, 7, 9	Temperature	T_ref	6.117

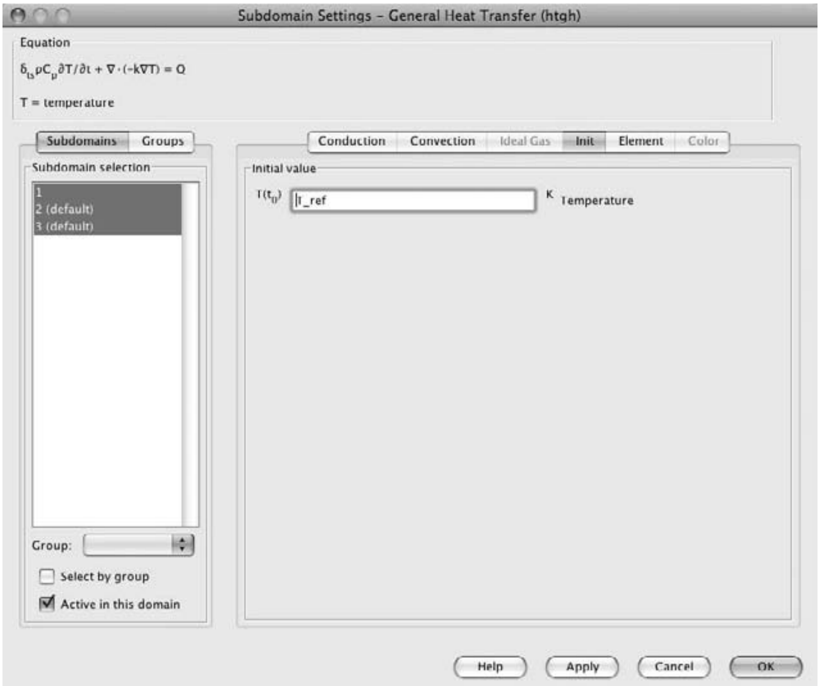


FIGURE 6.115 2D axisymmetric Inductive_Heating_1 model Subdomain Settings (1–3), Init edit window

FIGURE 6.116 2D axisymmetric Inductive_Heating_1 model Boundary Settings (1, 3, 5) edit window

Mesh Generation

On the toolbar, click the Initialize Mesh button once. This results in a mesh of approximately 1200 elements. See Figure 6.118.

Solving the 2D Axisymmetric Inductive_Heating_1 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Enter linspace(0,1200,21) in the Times edit window. See Figure 6.119.

Click the Advanced tab. Check the Use complex functions with real input check box. See Figure 6.120. Click OK.

FIGURE 6.117 2D axisymmetric Inductive_Heating_1 model Boundary Settings (2, 7, 9) edit window

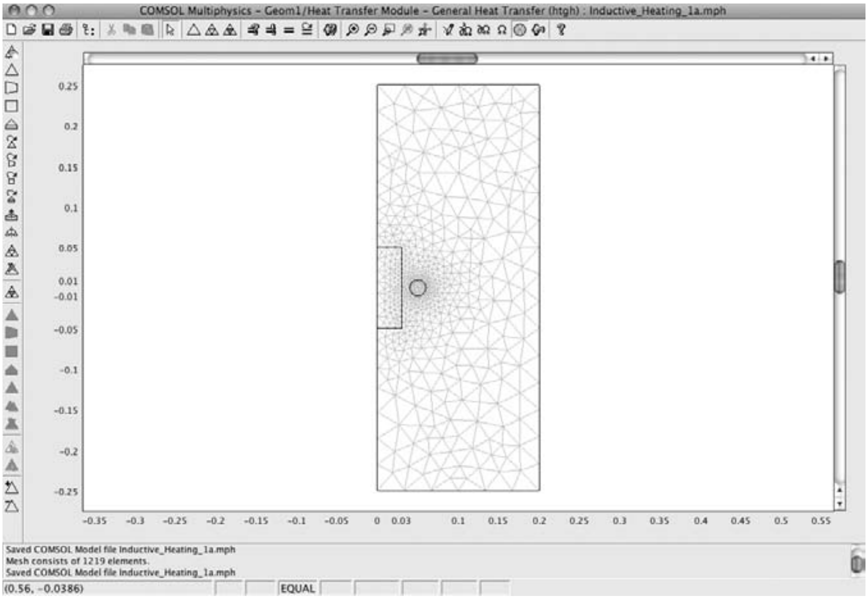


FIGURE 6.118 2D axisymmetric Inductive_Heating_1 model mesh window

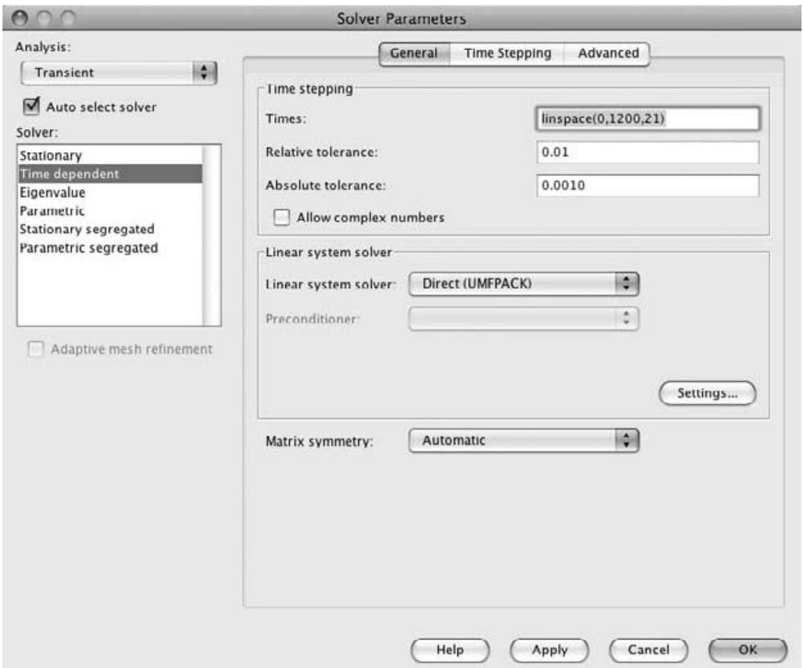


FIGURE 6.119 2D axisymmetric Inductive_Heating_1 model Solver Parameters edit window

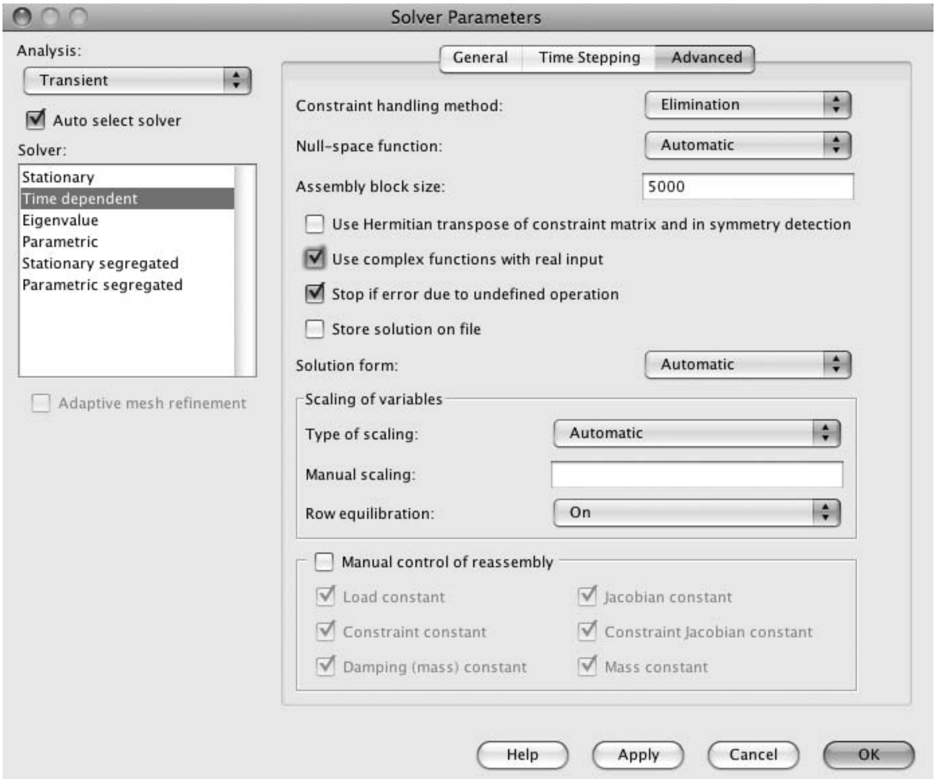


FIGURE 6.120 2D axisymmetric Inductive_Heating_1 model Solver Parameters, Advanced edit window

Time-Dependent Solving of the 2D Inductive_Heating_1 Model

Select Solve > Solve Problem. See Figure 6.121.

Postprocessing and Visualization

The default plot shows the temperature distribution in kelvins. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot Parameters > Surface. Verify that the Surface plot check box is checked and that the Predefined quantities pull-down list shows "Temperature." Select "degC" or "°C" from the Unit pull-down list. See Figure 6.122.

Click OK. See Figure 6.123.

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.124.

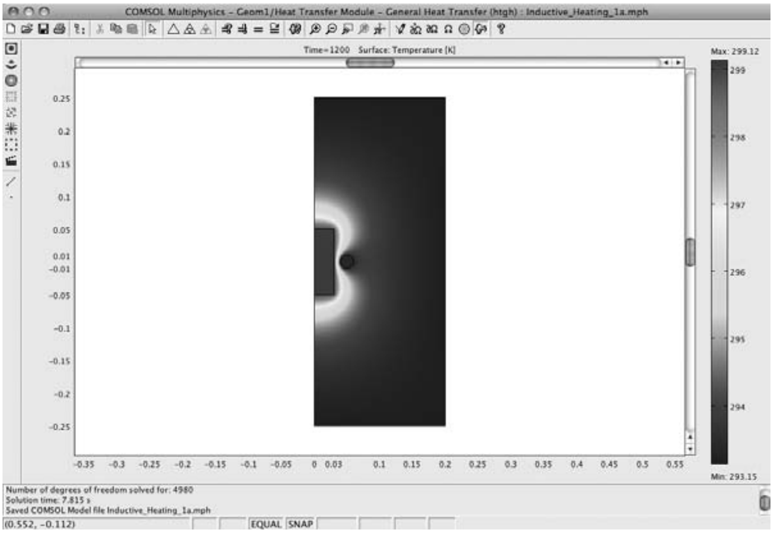


FIGURE 6.121 2D axisymmetric Inductive_Heating_1 model solution

FIGURE 6.122 2D axisymmetric Inductive_Heating_1 model Plot Parameters edit window

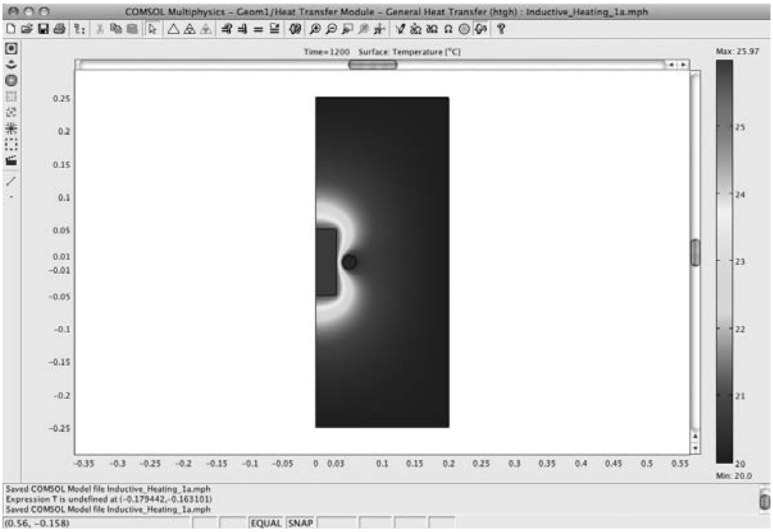


FIGURE 6.123 2D axisymmetric Inductive_Heating_1 model, degrees Centigrade

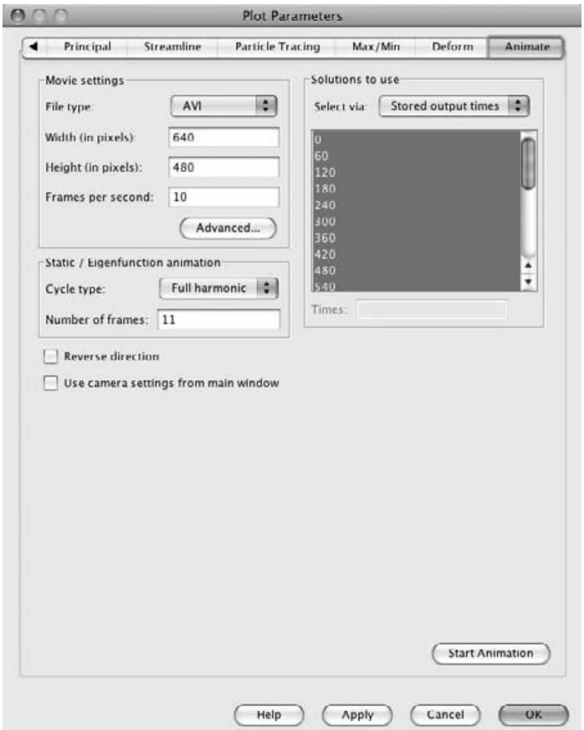


FIGURE 6.124 2D axisymmetric Inductive_Heating_1 model Plot Parameters edit window

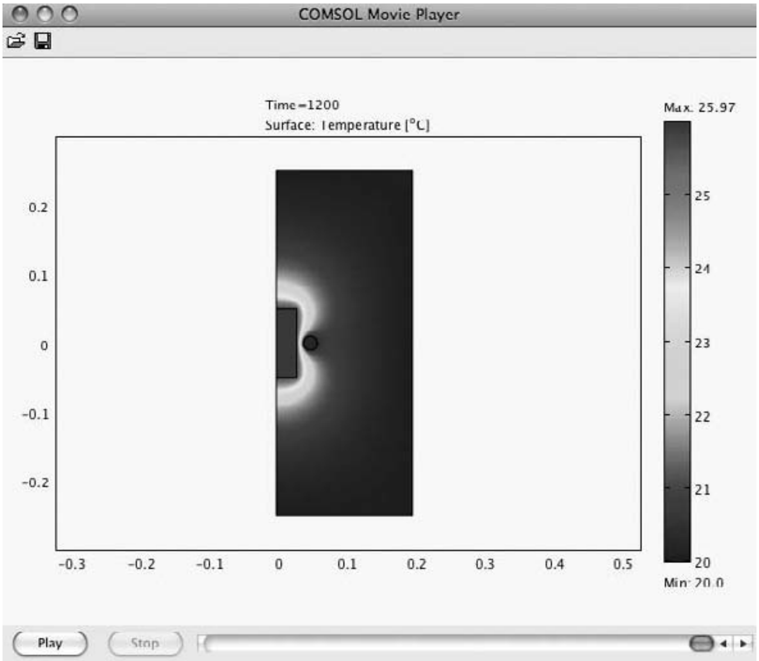


FIGURE 6.125 2D axisymmetric Inductive_Heating_1 model animation, final frame

Click the Start Animation button. See Figure 6.125.

Alternatively, you can play the file Movie6_IH_1.avi that was supplied with this book.

First Variation on the 2D Axisymmetric Inductive Heating Model

The following numerical solution model (Inductive_Heating_2) is derived from the Inductive_Heating_1 model. This variation continues the introduction of the coupling of two important basic physical materials properties—Joule heating and heat transfer—and expands on these concepts. The coupling of these two properties in this model demonstrates one of the applications normally found in typical engineering or process research.

NOTE This model is similar to the Inductive_Heating_1 model in the use of the induction heating method. However, in this case, the modeler will build an inductively heated crucible.

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat generation during a process and either adds heat or removes heat so as to achieve or

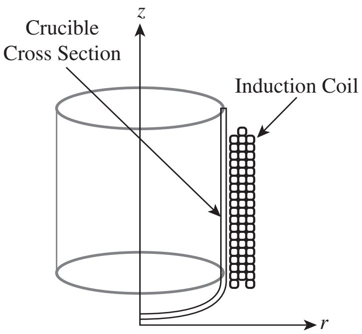


FIGURE 6.126 3D rendition of the 2D axisymmetric Inductive_Heating_2 model

maintain a desired temperature. Figure 6.126 shows a 3D rendition of the 2D axisymmetric Inductive_Heating_2 model geometry, as will be modeled here.

To start building the Inductive_Heating_2 Model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select AC/DC Module > Electro-Thermal Interaction > Azimuthal Induction Heating > Transient analysis. See Figure 6.127. Click OK.

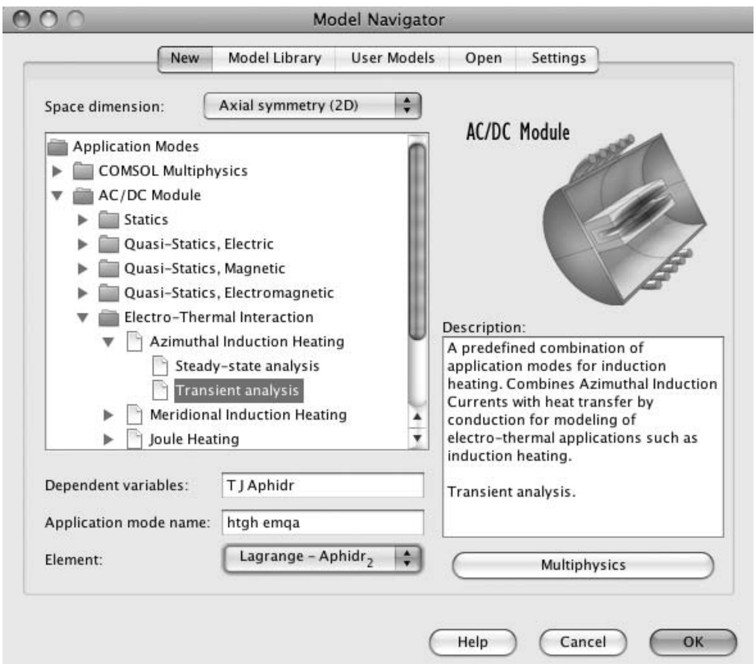


FIGURE 6.127 2D axisymmetric Inductive_Heating_2 Model Navigator setup

NOTE There are two mode names in the Application mode name window (htgh emqa). The names in the window show which applications are coupled in this application: General Heat Transfer (htgh) and Azimuthal Induction Currents, Vector Potential (emqa).

Options and Settings

NOTE When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

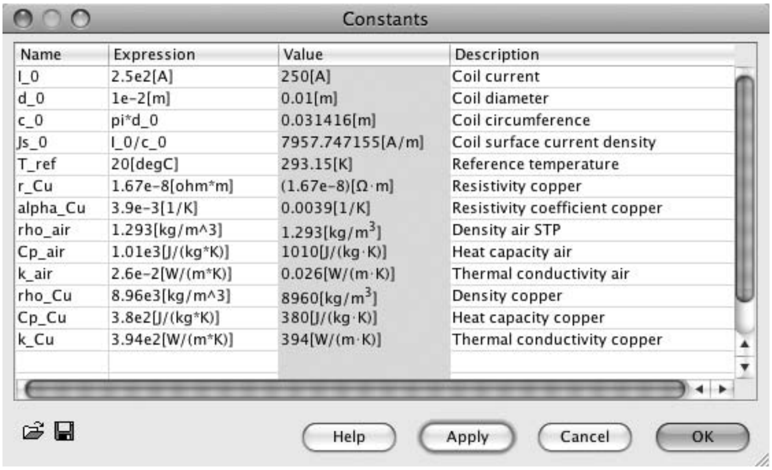
Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 6.32; also see Figure 6.128. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.2 and a height of 0.5. Select "Base: Corner" and set r equal to 0 and z equal to −0.1 in the Rectangle edit window. Click OK.

Table 6.32 Constants Edit Window

Name	Expression	Description
I_0	2.5e2[A]	Coil current
d_0	1e-2[m]	Coil diameter
c_0	pi*d_0	Coil circumference
Js_0	I_0/c_0	Coil surface current density
T_ref	20[degC]	Reference temperature
r_Cu	1.67e-8[ohm*m]	Resistivity copper
alpha_Cu	3.9e-3[1/K]	Resistivity coefficient copper
rho_air	1.293[kg/m^3]	Density air STP
Cp_air	1.01e3[J/(kg*K)]	Heat capacity air
k_air	2.6e-2[W/(m*K)]	Thermal conductivity air
rho_Cu	8.96e3[kg/m^3]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity copper
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity copper



Name	Expression	Value	Description
I_0	2.5e2[A]	250[A]	Coil current
d_0	1e-2[m]	0.01[m]	Coil diameter
c_0	pi*d_0	0.031416[m]	Coil circumference
Js_0	I_0/c_0	7957.747155[A/m]	Coil surface current density
T_ref	20[degC]	293.15[K]	Reference temperature
r_Cu	1.67e-8[ohm*m]	(1.67e-8)[Ω·m]	Resistivity copper
alpha_Cu	3.9e-3[1/K]	0.0039[1/K]	Resistivity coefficient copper
rho_air	1.293[kg/m^3]	1.293[kg/m³]	Density air STP
Cp_air	1.01e3[J/(kg*K)]	1010[J/(kg·K)]	Heat capacity air
k_air	2.6e-2[W/(m*K)]	0.026[W/(m·K)]	Thermal conductivity air
rho_Cu	8.96e3[kg/m^3]	8960[kg/m³]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	380[J/(kg·K)]	Heat capacity copper
k_Cu	3.94e2[W/(m*K)]	394[W/(m·K)]	Thermal conductivity copper

FIGURE 6.128 2D axisymmetric Inductive_Heating_2 model Constants edit window

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.1 and a height of 0.25. Select “Base: Corner” and set r equal to 0 and z equal to 0.05 in the Rectangle edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes of 0.1 and B-semiaxes of 0.05. Select “Base: Center” and set r equal to 0 and z equal to 0.05 in the Ellipse edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.1 and a height of 0.30. Select “Base: Corner” and set r equal to −0.1 and z equal to 0 in the Rectangle edit window. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter E1−R3 in the Set formula edit window. See Figure 6.129. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter R2+CO1 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 6.130.

Click OK. See Figure 6.131.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter width of 0.1–0.005 and a height of 0.25. Select “Base: Corner” and set r equal to 0 and z equal to 0.05 in the Rectangle edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes of 0.1–0.005 and B-semiaxes of 0.05–0.005. Select “Base: Corner” and set r equal to 0 and z equal to 0.05 in the Ellipse edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.1 and a height of 0.30. Select “Base: Corner” and set r equal to −0.1 and z equal to 0 in the Rectangle edit window. Click OK.

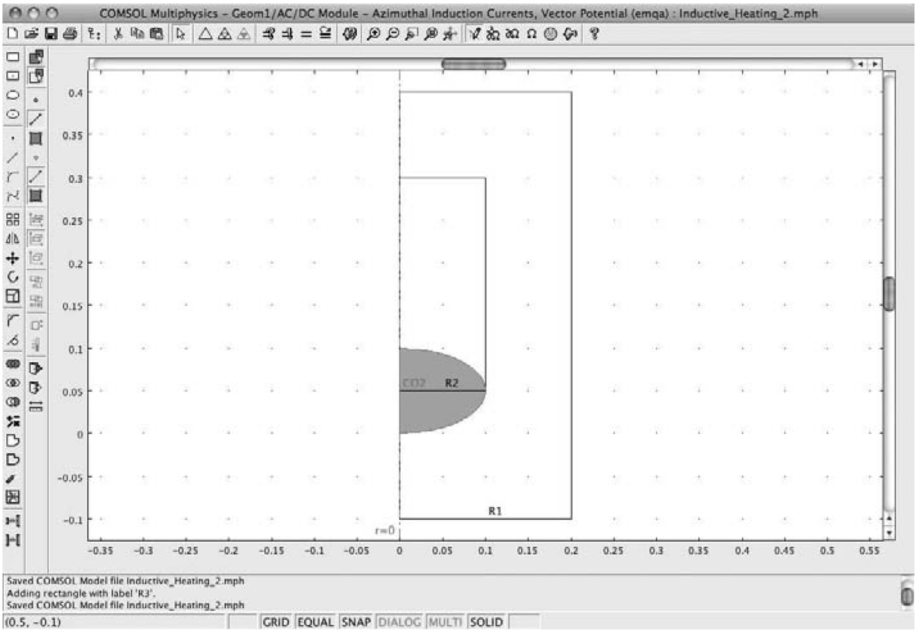


FIGURE 6.129 2D axisymmetric Inductive_Heating_2 model Create Composite Object edit window

Using the menu bar, select Draw > Create Composite Object. Enter E1−R3 in the Set formula edit window. See Figure 6.132. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter R2+CO2 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 6.133.

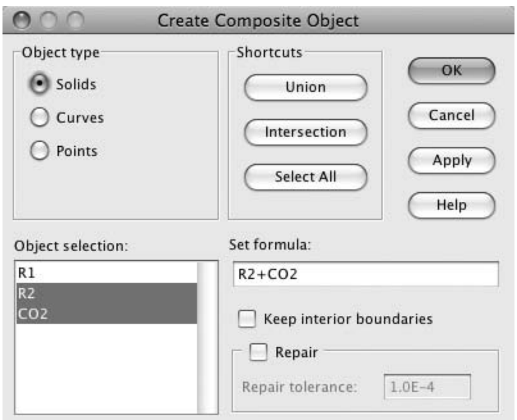


FIGURE 6.130 2D axisymmetric Inductive_Heating_2 model Create Composite Object edit window

FIGURE 6.131 2D axisymmetric Inductive_Heating_2 model (CO2)

Click OK. See Figure 6.134.

Using the menu bar, select Draw > Create Composite Object. Enter: CO1−CO3 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 6.135.

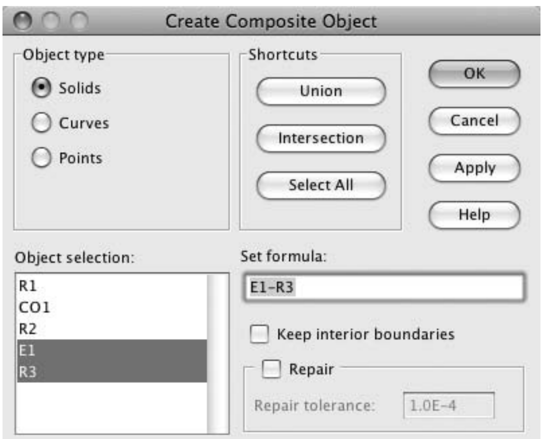


FIGURE 6.132 2D axisymmetric Inductive_Heating_2 model Create Composite Object edit window

FIGURE 6.133 2D axisymmetric Inductive_Heating_2 model Create Composite Object edit window

FIGURE 6.134 2D axisymmetric Inductive_Heating_2 model (CO1, CO3)

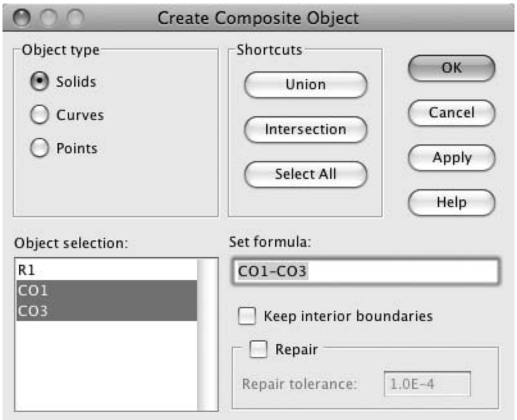


FIGURE 6.135 2D axisymmetric Inductive_Heating_2 model Create Composite Object edit window

Click OK. See Figure 6.136.

Having built the crucible, the next step is to build the first layer of the heating coil. Using the menu bar, select Draw > Specify Objects > Circle. Create each of the circles shown in Table 6.33.

FIGURE 6.136 2D axisymmetric Inductive_Heating_2 model, crucible

Table 6.33 Circle Edit Window

Name	Radius	Base	*r*	*z*
C1	0.005	Center	0.11	0.260
C2	0.005	Center	0.11	0.248
C3	0.005	Center	0.11	0.236
C4	0.005	Center	0.11	0.224
C5	0.005	Center	0.11	0.212
C6	0.005	Center	0.11	0.200
C7	0.005	Center	0.11	0.188
C8	0.005	Center	0.11	0.176
C9	0.005	Center	0.11	0.164
C10	0.005	Center	0.11	0.152
C11	0.005	Center	0.11	0.140
C12	0.005	Center	0.11	0.128
C13	0.005	Center	0.11	0.116
C14	0.005	Center	0.11	0.104
C15	0.005	Center	0.11	0.092
C16	0.005	Center	0.11	0.080

Building the second layer of the heating coil is easier than building the first layer. Select circles C1–C16. Using the menu bar, select Edit > Copy. Using the menu bar, select Edit > Paste. Enter r = 0.01 and z = −0.006. Click OK.

Using the menu bar, select Edit > Paste. Enter r = 0.02 and z = 0. Click OK. See Figure 6.137.

Physics Settings

Select Physics > Scalar Variables. Enter 500 in the nu_emqa edit window. See Figure 6.138. Click OK.

Select Options > Expressions > Scalar Expressions. Enter the scalar expression for sigma_Cu_T as shown in Table 6.34; also see Figure 6.139. Click OK.

NOTE The scalar expression for sigma_Cu_T couples the resistivity of copper (r_Cu) and the temperature (T).

The final geometric step is to create a composite object. Using the menu bar, select Draw > Create Composite Object. Select all of the objects. Check the Keep interior boundaries check box. See Figure 6.140.

FIGURE 6.137 2D axisymmetric Inductive_Heating_2 model, crucible and coil

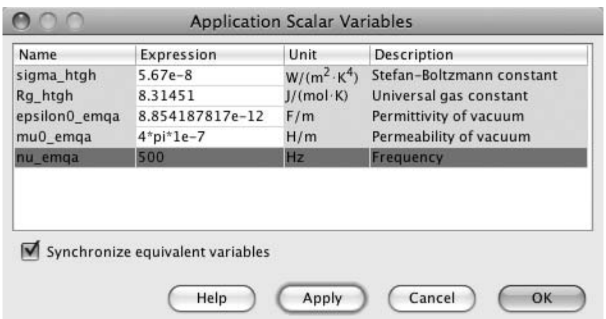


FIGURE 6.138 2D axisymmetric Inductive_Heating_2 model Application Scalar Variables edit window

Table 6.34 Scalar Expressions Edit Window

Name	Expression	Description
sigma_Cu_T	1/(r_Cu*+(1 + alpha_Cu*(T-T_ref)))	Electrical conductivity, Cu

Click OK. See Figure 6.141.

Now that the 2D axisymmetric Inductive_Heating_2 model geometry is built, export it to a file for future use. Select File > Export > Geometry Objects to File. Enter IH_2_Geometry in the Save as window. Select "DXF File" from the File Format pull-down list. Click OK.

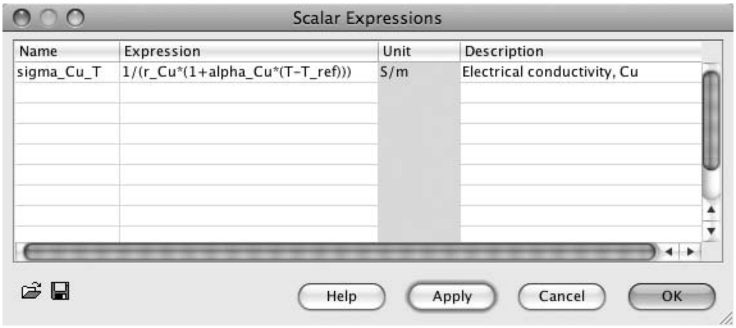


FIGURE 6.139 2D axisymmetric Inductive_Heating_2 model Scalar Expressions edit window

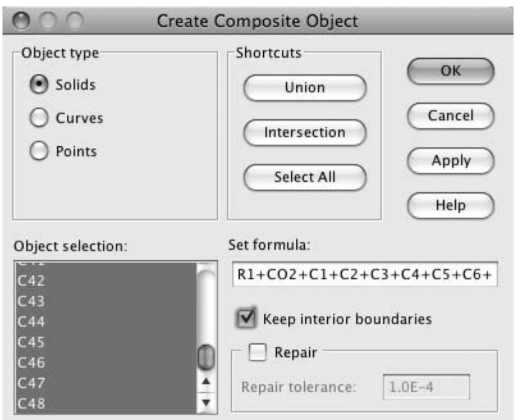


FIGURE 6.140 2D axisymmetric Inductive_Heating_2 model Create Composite Object edit window

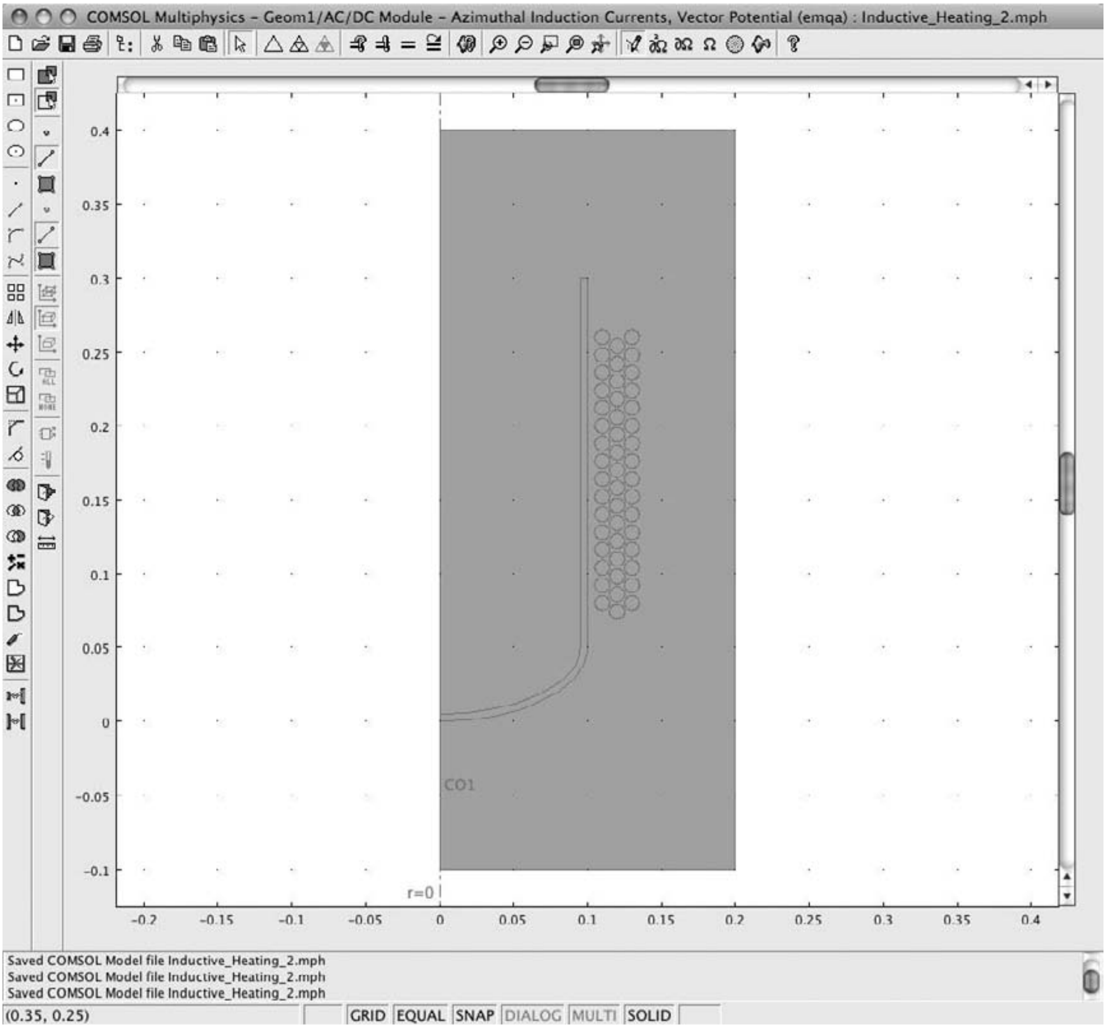


FIGURE 6.141 2D axisymmetric Inductive_Heating_2 model

Physics Subdomain Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Multiphysics > Azimuthal Induction Currents, Vector Potential (emqa). Using the menu bar, select Physics > Subdomain Settings > Electric Parameters. In the Subdomain edit windows, enter the information shown in Table 6.35. Click OK. See Figures 6.142, 6.143, and 6.144.

Table 6.35 Subdomain Edit Window

Subdomain	Name	Expression	Description	Figure Number
1	σ	0	Electric conductivity	6.142
2	σ	sigma_Cu_T	Electric conductivity	6.143
3–50	σ	0	Electric conductivity	6.144

Physics Boundary Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.36. Check the Interior boundaries check box. See Figures 6.145 and 6.146.

Table 6.36 Boundary Settings Edit Window

Boundary	Boundary Condition	Figure Number
1, 3, 4	Axial symmetry	6.145
2, 5, 9	Magnetic insulation	6.146

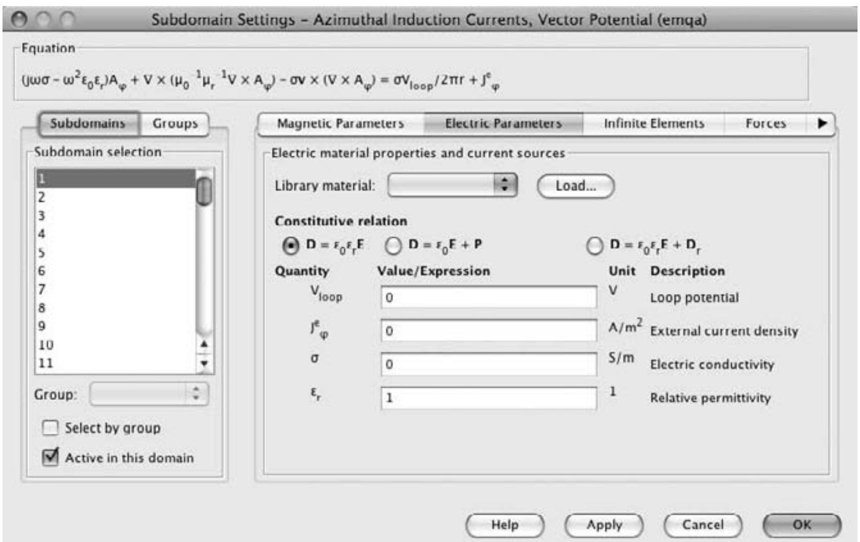


FIGURE 6.142 2D axisymmetric Inductive_Heating_2 model Subdomain Settings (1), Electric Parameters edit window

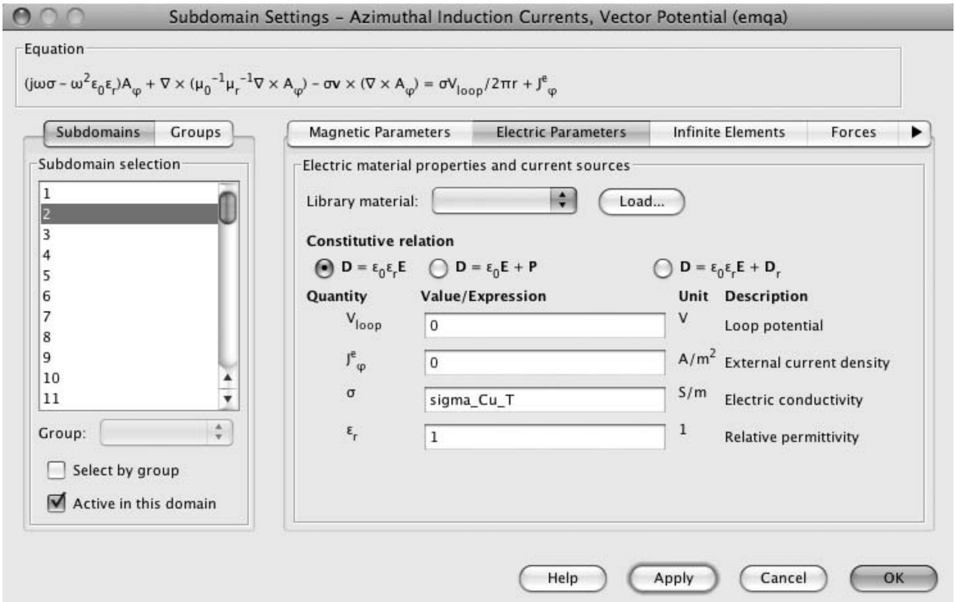


FIGURE 6.143 2D axisymmetric Inductive_Heating_2 model Subdomain Settings (2), Electric Parameters edit window

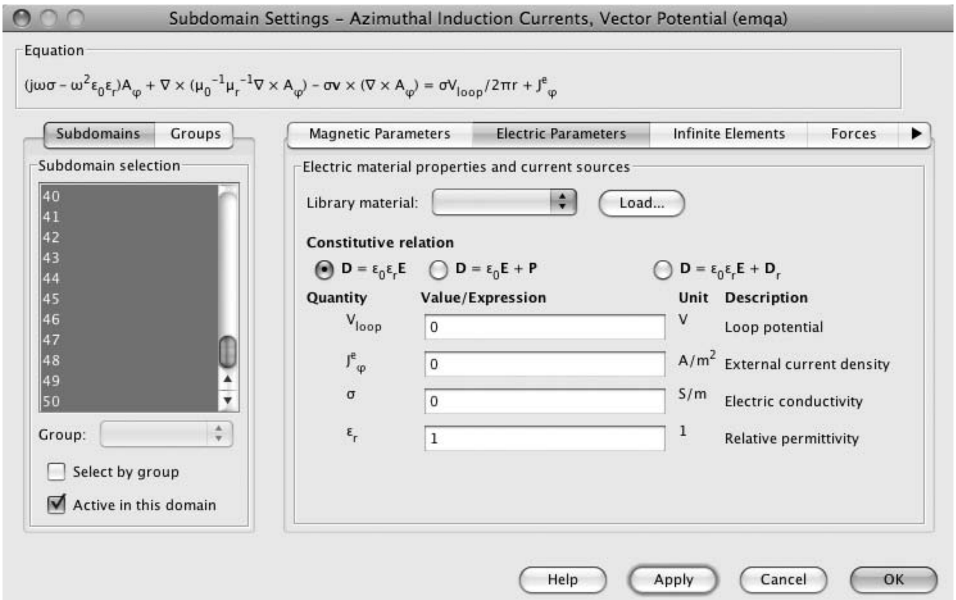


FIGURE 6.144 2D axisymmetric Inductive_Heating_2 model Subdomain Settings (3–50), Electric Parameters edit window

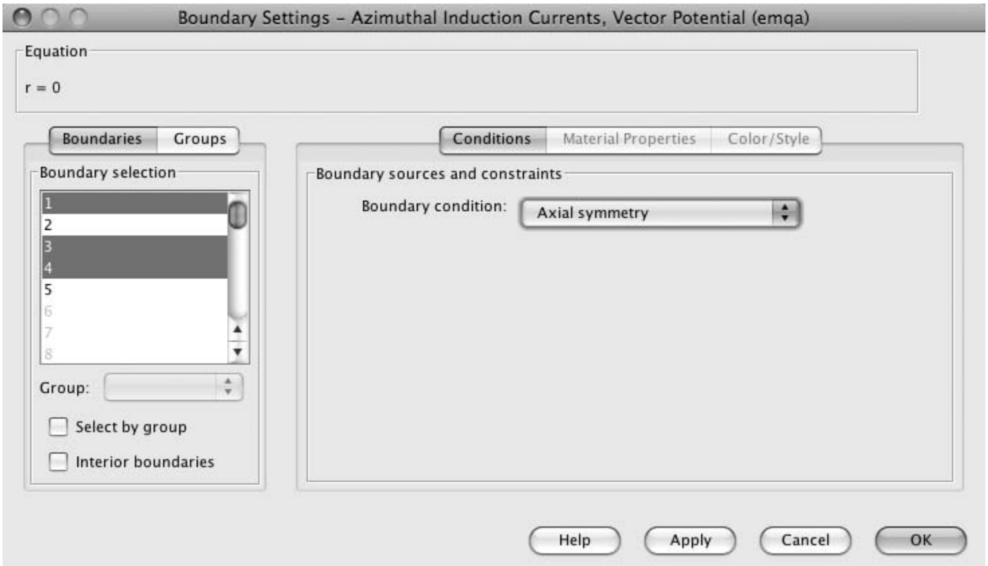


FIGURE 6.145 2D axisymmetric Inductive_Heating_2 model Boundary Settings (1, 3, 4) edit window

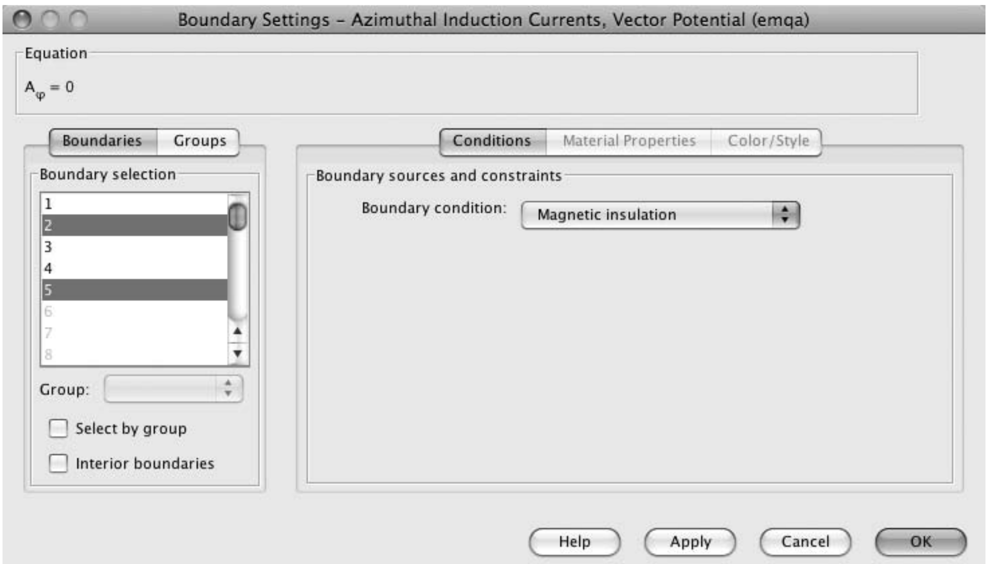


FIGURE 6.146 2D axisymmetric Inductive_Heating_2 model Boundary Settings (2, 5, 9) edit window

Table 6.37 Boundary Settings Edit Window

Boundary	Boundary Condition	Value	Figure Number
12–203	Surface current	Js_0	6.147

FIGURE 6.147 2D axisymmetric Inductive_Heating_2 model Boundary Settings (12–203) edit window

In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.37. See Figure 6.147.

Click OK. See Figure 6.148.

Physics Subdomain Settings: General Heat Transfer (htgh)

Using the menu bar, select Multiphysics > General Heat Transfer (htgh). Using the menu bar, select Physics > Subdomain Settings > Conduction. In the Subdomain edit windows, enter the information shown in Table 6.38. See Figures 6.149 and 6.150.

Table 6.38 Subdomain Edit Window

Subdomain	Name	Expression	Description	Figure Number
1	*k* (isotropic)	k_air	Thermal conductivity	6.149
	ρ	rho_air	Density	
	*C*p	Cp_air	Heat capacity	
2–50	*k* (isotropic)	k_Cu	Thermal conductivity	6.150
	ρ	rho_Cu	Density	
	*C*p	Cp_Cu	Heat capacity	

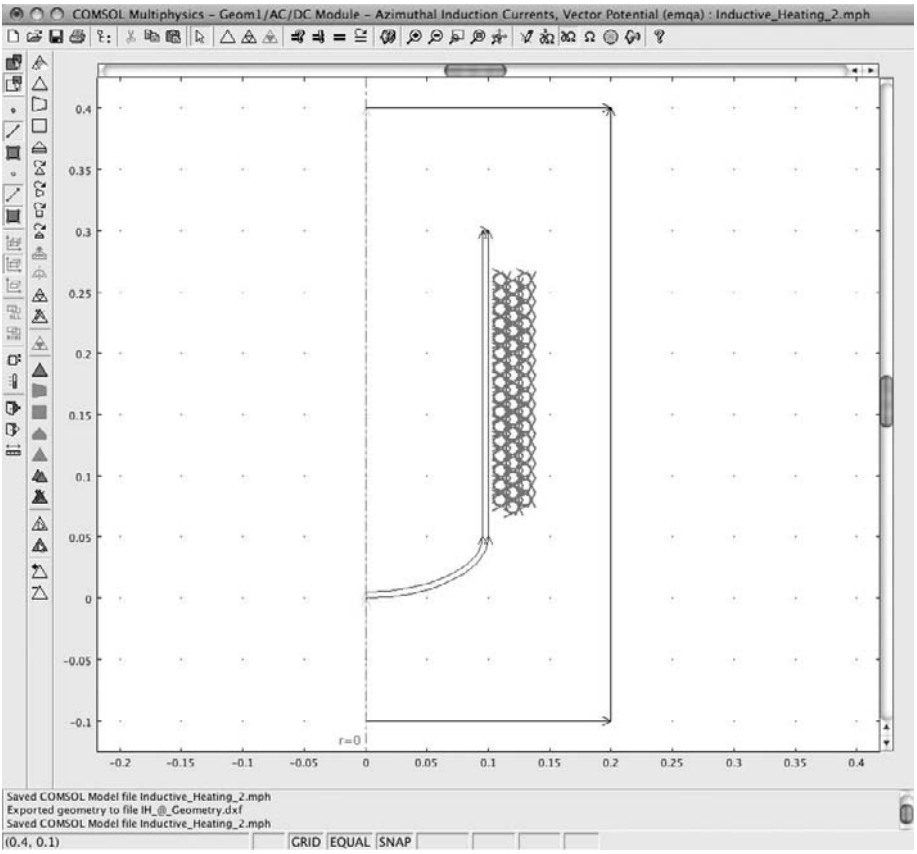


FIGURE 6.148 2D axisymmetric Inductive_Heating_2 model, coil

Select the Init tab. Select subdomains 1–50. Enter T_ref in the $T(t_0)$ edit window. See Figure 6.151. Click OK.

Physics Boundary Settings: General Heat Transfer (htgh)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – General Heat Transfer (htgh) edit windows, enter the information shown in Table 6.39. Click OK. See Figures 6.152 and 6.153.

Table 6.39 Boundary Settings Edit Window

Boundary	Boundary Condition	Value	Figure Number
1, 3, 4	Axial symmetry	—	6.152
2, 5, 9	Temperature	T_ref	6.153

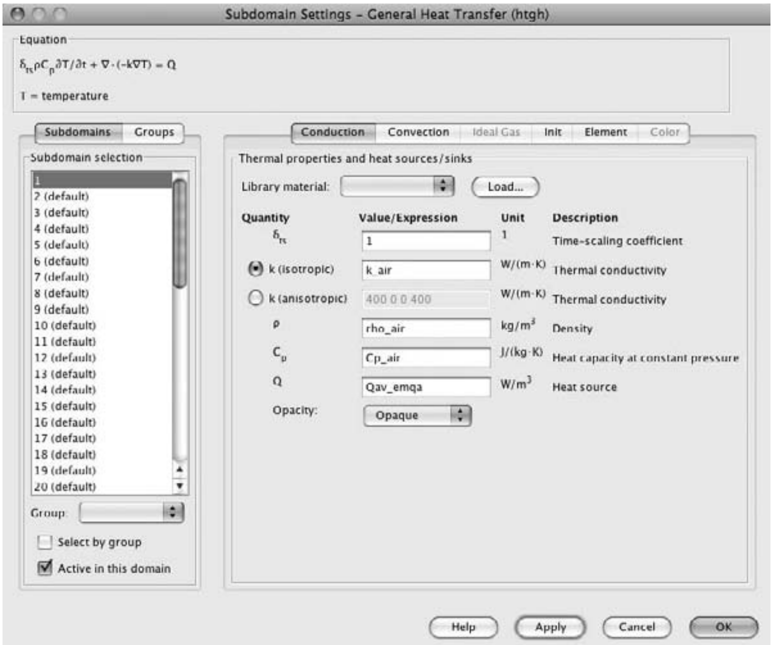


FIGURE 6.149 2D axisymmetric Inductive_Heating_2 model Subdomain Settings (1) edit window

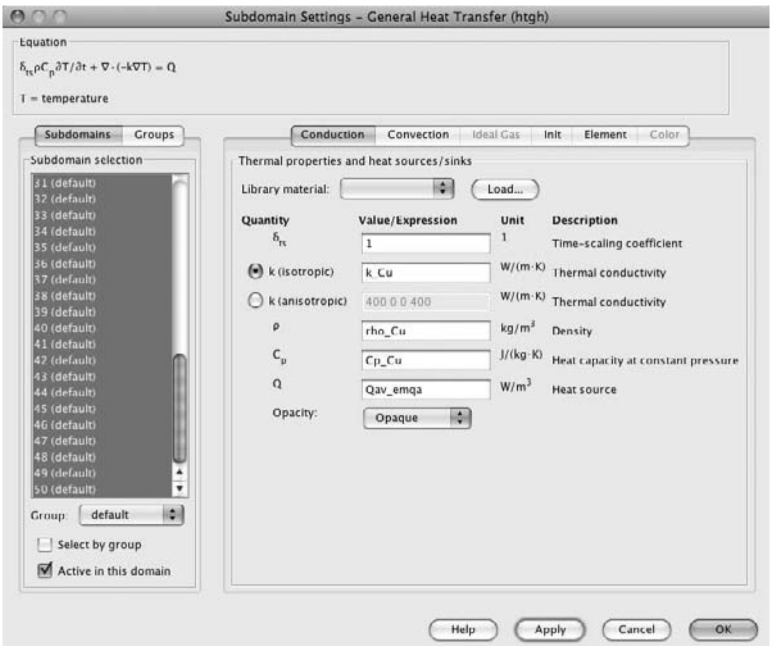


FIGURE 6.150 2D axisymmetric Inductive_Heating_2 model Subdomain Settings (2–50) edit window

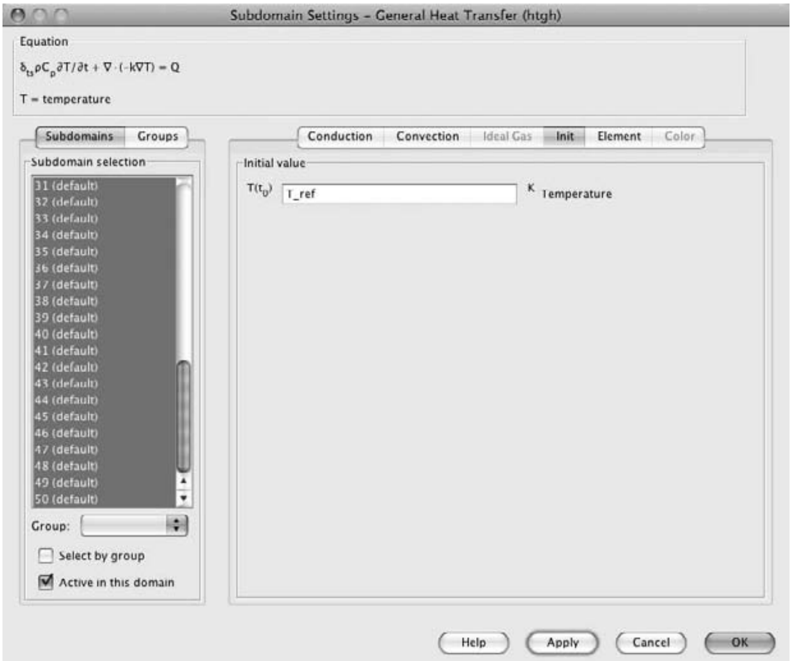


FIGURE 6.151 2D axisymmetric Inductive_Heating_2 model Subdomain Settings (1–50), Init edit window

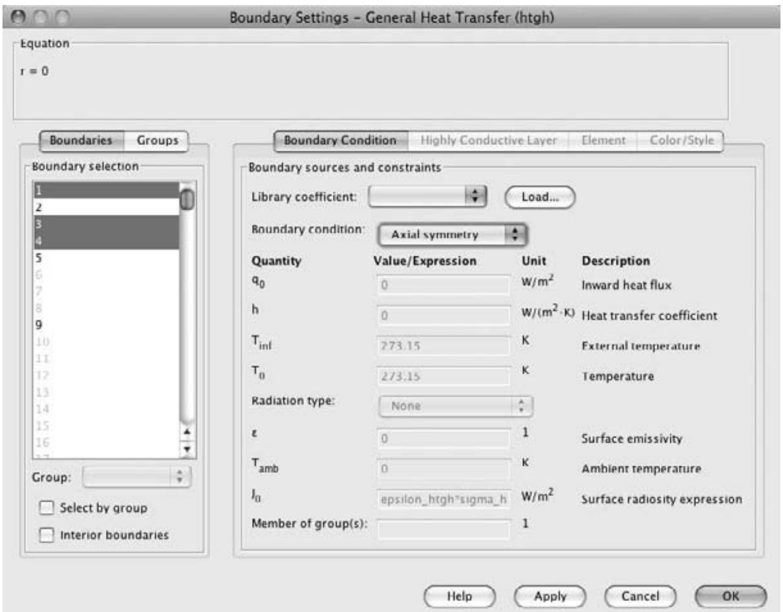


FIGURE 6.152 2D axisymmetric Inductive_Heating_2 model Boundary Settings (1, 3, 4) edit window

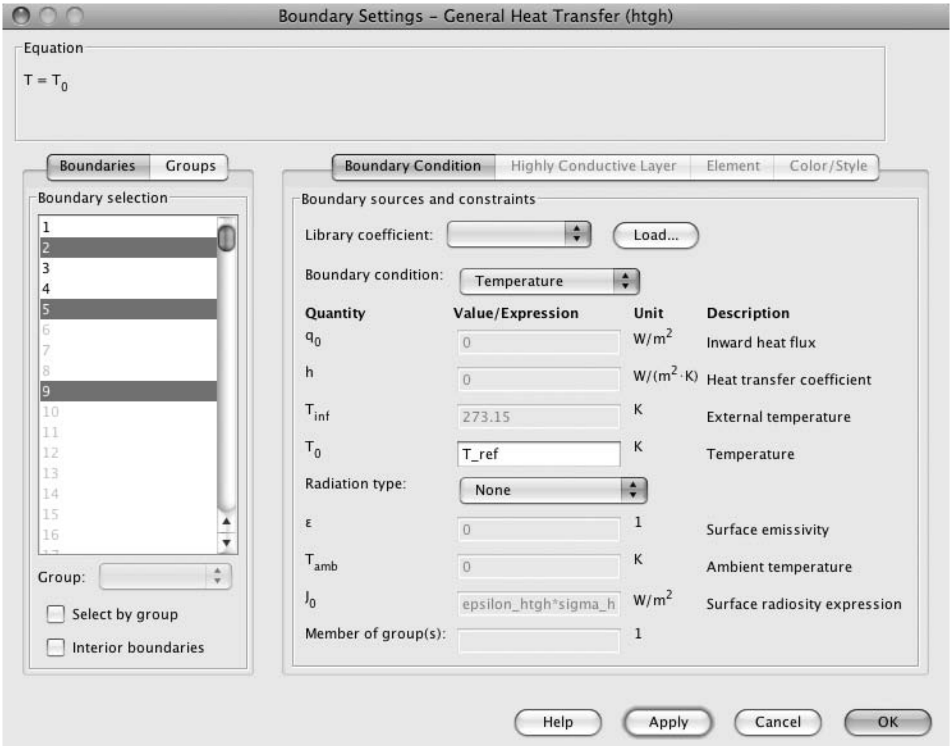


FIGURE 6.153 2D axisymmetric Inductive_Heating_2 model Boundary Settings (2, 5, 9) edit window

Mesh Generation

On the toolbar, click the Initialize Mesh button once. This mesh yields approximately 14,000 elements. See Figure 6.154.

Solving the 2D Axisymmetric Inductive_Heating_2 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

FIGURE 6.154 2D axisymmetric Inductive_Heating_2 model mesh window

Enter linspace(0,1200,21) in the Times edit window. See Figure 6.155.

Click the Advanced tab. Check the Use complex functions with real input check box. See Figure 6.156. Click OK.

Time-Dependent Solving of the 2D Resistive_Heating_2 Model

Select Solve > Solve Problem. See Figure 6.157.

Postprocessing and Visualization

The default plot shows the temperature distribution in kelvins. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot Parameters > Surface. Verify that the Surface plot check box is checked and that

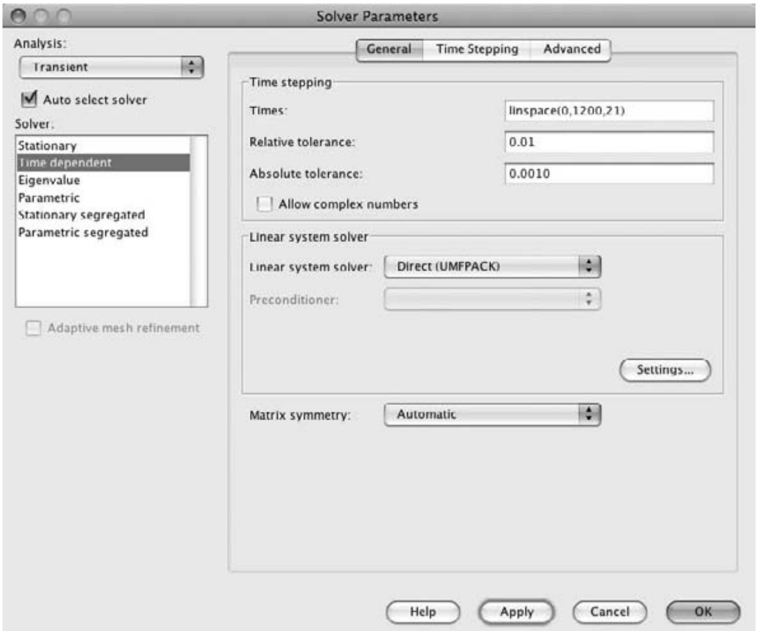


FIGURE 6.155 2D axisymmetric Inductive_Heating_2 model Solver Parameters edit window

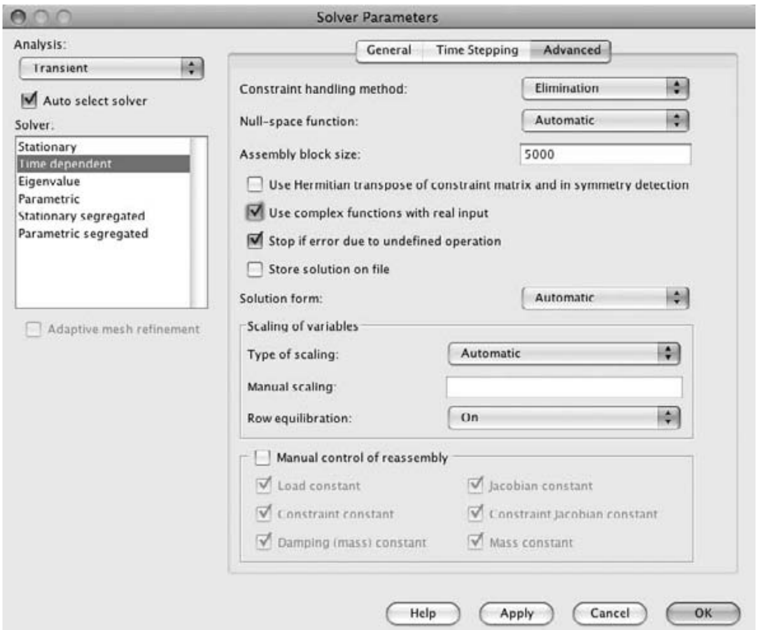


FIGURE 6.156 2D axisymmetric Inductive_Heating_2 model Solver Parameters, Advanced edit window

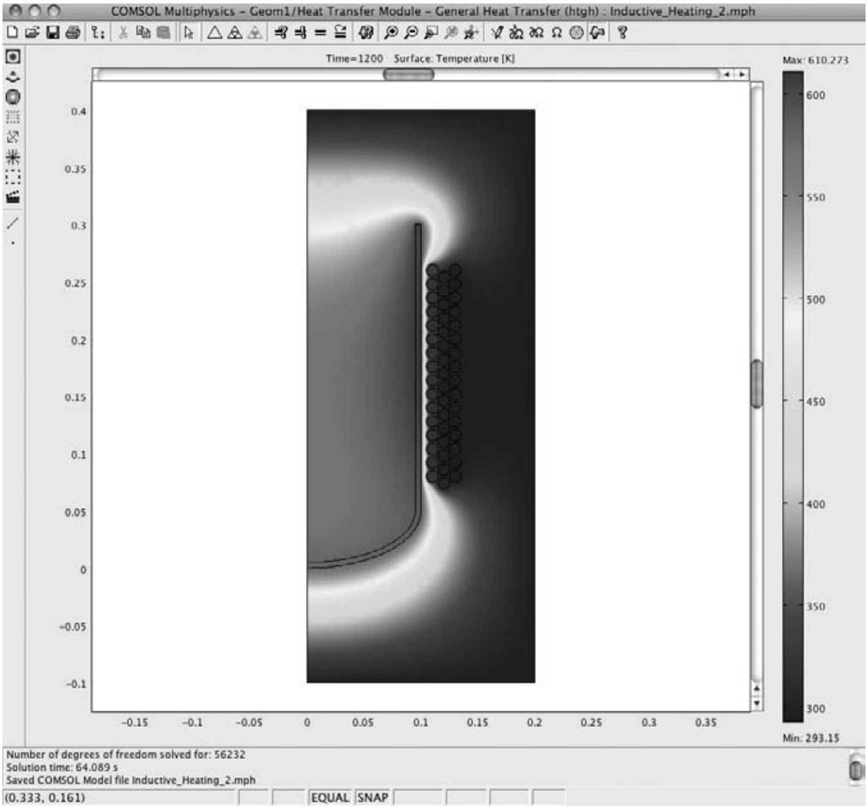


FIGURE 6.157 2D axisymmetric Inductive_Heating_2 model solution

the Predefined quantities pull-down list shows "Temperature." Select "degC" or "°C" from the Unit pull-down list. See Figure 6.158.

Click OK. See Figure 6.159.

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.160.

Click the Start Animation button. See Figure 6.161.

Alternatively, you can play the file Movie6_IH_2.avi that was supplied with this book.

Second Variation on the 2D Axisymmetric Inductive Heating Model

The following numerical solution model (Inductive_Heating_3) is derived from the Inductive_Heating_2 model. It continues the introduction of the coupling of two

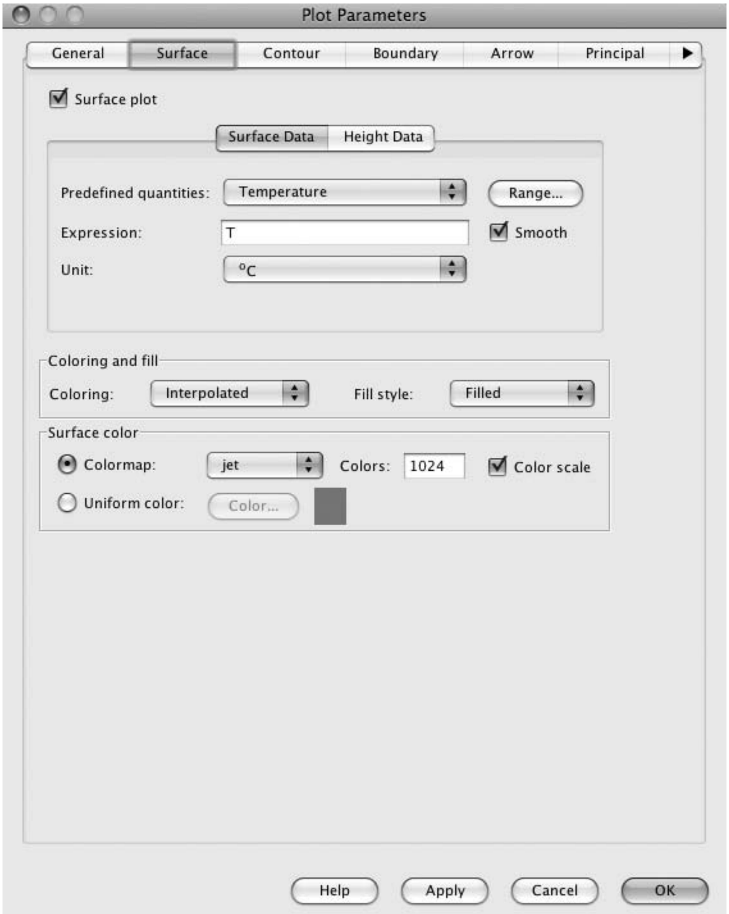


FIGURE 6.158 2D axisymmetric Inductive_Heating_2 model Plot Parameters edit window

important basic physical materials properties—Joule heating and heat transfer—and expands on these concepts. The coupling of these two properties in this model demonstrates one of the applications normally found in typical engineering or process research.

NOTE This model is similar to the Inductive_Heating_2 model in the use of the induction heating method. However, in this case, the modeler will build a filled (loaded) inductively heated crucible. Here, bismuth[21,22] is the material of choice. Additionally, the crucible will be surrounded by nitrogen[23] to prevent oxidation of the heated bismuth.

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat

FIGURE 6.159 2D axisymmetric Inductive_Heating_2 model, degrees Centigrade

generation during a process and either adds heat or removes heat so as to achieve or maintain a desired temperature. Figure 6.162 shows a 3D rendition of the 2D axisymmetric Inductive_Heating_3 model geometry, as will be modeled here.

To start building the Inductive_Heating_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select “Axial symmetry (2D)” from the Space dimension pull-down list. Select AC/DC Module > Electro-Thermal Interaction > Azimuthal Induction Heating > Transient analysis. See Figure 6.163. Click OK.

NOTE There are two mode names in the Application mode name window (htgh emqa). The names in the window show which applications are coupled in this application: General Heat Transfer (htgh) and Azimuthal Induction Currents, Vector Potential (emqa).

FIGURE 6.160 2D axisymmetric Inductive_Heating_2 model Plot Parameters window

FIGURE 6.161 2D axisymmetric Inductive_Heating_2 model animation, final frame

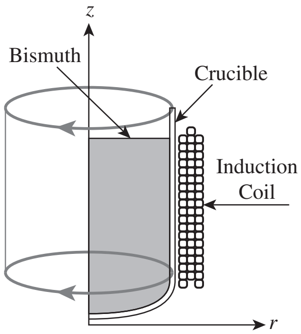


FIGURE 6.162 3D rendition of the 2D axisymmetric Inductive_Heating_3 model

Options and Settings

Constants

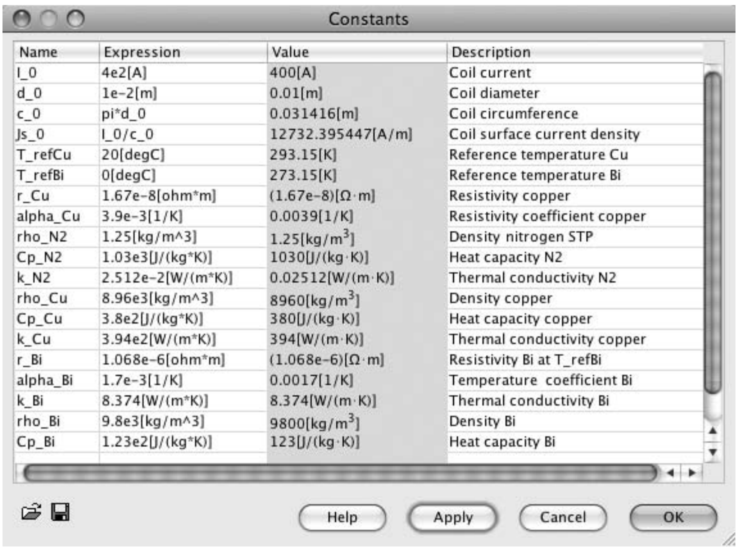
Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 6.40; see also Figure 6.164. Click OK.

Select File > Import > CAD Data From File > IH_2_Geometry.dxf. Click the Import button.

FIGURE 6.163 2D axisymmetric Inductive_Heating_3 Model Navigator setup

Table 6.40 Constants Edit Window

Name	Expression	Description
I_0	4e2[A]	Coil current
d_0	1e-2[m]	Coil diameter
c_0	pi*d_0	Coil circumference
Js_0	I_0/c_0	Coil surface current density
T_refCu	20[degC]	Reference temperature Cu
T_refBi	0[degC]	Reference temperature Bi
r_Cu	1.67e-8[ohm*m]	Resistivity copper
alpha_Cu	3.9e-3[1/K]	Resistivity coefficient copper
rho_N2	1.25[kg/m^3]	Density nitrogen STP
Cp_N2	1.03e3[J/(kg*K)]	Heat capacity N2
k_N2	2.512e-2[W/(m*K)]	Thermal conductivity N2
rho_Cu	8.96e3[kg/m^3]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity copper
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity copper
r_Bi	1.068e-6[ohm*m]	Resistivity Bi at T_refBi
alpha_Bi	1.7e-3[1/K]	Temperature coefficient Bi
k_Bi	8.374[W/(m*K)]	Thermal conductivity Bi
rho_Bi	9.8e3[kg/m^3]	Density Bi
Cp_Bi	1.23e2[J/(kg*K)]	Heat capacity Bi



Constants

Name	Expression	Value	Description
I_0	4e2[A]	400[A]	Coil current
d_0	1e-2[m]	0.01[m]	Coil diameter
c_0	pi*d_0	0.031416[m]	Coil circumference
Js_0	I_0/c_0	12732.395447[A/m]	Coil surface current density
T_refCu	20[degC]	293.15[K]	Reference temperature Cu
T_refBi	0[degC]	273.15[K]	Reference temperature Bi
r_Cu	1.67e-8[ohm*m]	(1.67e-8)[Ω·m]	Resistivity copper
alpha_Cu	3.9e-3[1/K]	0.0039[1/K]	Resistivity coefficient copper
rho_N2	1.25[kg/m^3]	1.25[kg/m^3]	Density nitrogen STP
Cp_N2	1.03e3[J/(kg*K)]	1030[J/(kg·K)]	Heat capacity N2
k_N2	2.512e-2[W/(m*K)]	0.02512[W/(m·K)]	Thermal conductivity N2
rho_Cu	8.96e3[kg/m^3]	8960[kg/m^3]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	380[J/(kg·K)]	Heat capacity copper
k_Cu	3.94e2[W/(m*K)]	394[W/(m·K)]	Thermal conductivity copper
r_Bi	1.068e-6[ohm*m]	(1.068e-6)[Ω·m]	Resistivity Bi at T_refBi
alpha_Bi	1.7e-3[1/K]	0.0017[1/K]	Temperature coefficient Bi
k_Bi	8.374[W/(m*K)]	8.374[W/(m·K)]	Thermal conductivity Bi
rho_Bi	9.8e3[kg/m^3]	9800[kg/m^3]	Density Bi
Cp_Bi	1.23e2[J/(kg*K)]	123[J/(kg·K)]	Heat capacity Bi

Help Apply Cancel OK

FIGURE 6.164 2D axisymmetric Inductive_Heating_3 model Constants edit window

FIGURE 6.165 2D axisymmetric Inductive_Heating_3 model, imported crucible and coil with rectangle

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.2 and a height of 0.5. Select “Base: Corner” and set r equal to 0 and z equal to -0.1 in the Rectangle edit window. Click OK. See Figure 6.165.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.1–0.005 and a height of 0.25–0.05. Select “Base: Corner” and set r equal to 0 and z equal to 0.05 in the Rectangle edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes of 0.1–0.005 and B-semiaxes of 0.05–0.005. Select “Base: Corner” and set r equal to 0 and z equal to 0.05 in the Ellipse edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.1 and a height of 0.30. Select “Base: Corner” and set r equal to -0.1 and z equal to 0 in the Rectangle edit window. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter E1−R3 in the Set formula edit window. Click OK.

FIGURE 6.166 2D axisymmetric Inductive_Heating_3 model, bismuth subdomain 3

Using the menu bar, select Draw > Create Composite Object. Enter R2+CO1 in the Set formula edit window. Verify that the Keep interior boundaries check box is not checked. Click OK.

Select Physics > Subdomain Settings. Select subdomain 3 to verify that the bismuth subdomain has been properly added. Click OK. See Figure 6.166.

Physics Settings

Select Physics > Scalar Variables. Enter 500 in the nu_emqa edit window. See Figure 6.167. Click OK.

Select Options > Expressions > Scalar Expressions. Enter the scalar expressions for sigma_Cu_T and sigma_Bi_T as shown in Table 6.41; also see Figure 6.168. Click OK.

Table 6.41 Scalar Expressions Edit Window

Name	Expression	Description
sigma_Cu_T	1/(r_Cu*(1+alpha_Cu*(T-T_refCu)))	Electrical conductivity, Cu
sigma_Bi_T	1/(r_Bi*(1+alpha_Bi*(T-T_refBi)))	Electrical conductivity, Bi

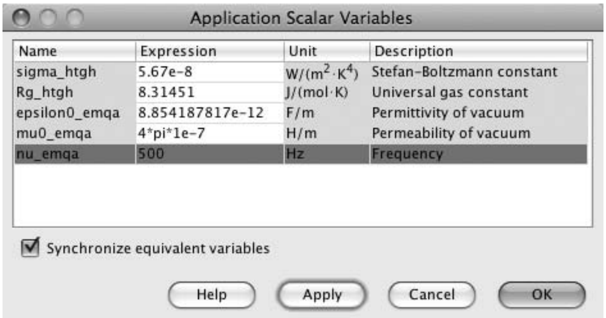


FIGURE 6.167 2D axisymmetric Inductive_Heating_3 model Application Scalar Variables edit window

NOTE The scalar expression for sigma_Cu_T couples the resistivity of copper (r_Cu) and the temperature (T). The scalar expression for sigma_Bi_T couples the resistivity of bismuth (r_Bi) and the temperature (T).

Physics Subdomain Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Multiphysics > Azimuthal Induction Currents, Vector Potential (emqa). Using the menu bar, select Physics > Subdomain Settings > Electric

FIGURE 6.168 2D axisymmetric Inductive_Heating_3 model Scalar Expressions edit window

Table 6.42 Subdomain Edit Window

Subdomain	Name	Expression	Description	Figure Number
1	σ	0	Electric conductivity	6.169
2	σ	sigma_Cu_T	Electric conductivity	6.170
3	σ	sigma_Bi_T	Electric conductivity	6.171
4–51	σ	0	Electric conductivity	6.172

Parameters. In the Subdomain edit windows, enter the information shown in Table 6.42. Click OK. See Figures 6.169–6.172.

Physics Boundary Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.43. Check the Interior boundaries check box. See Figures 6.173 and 6.174.

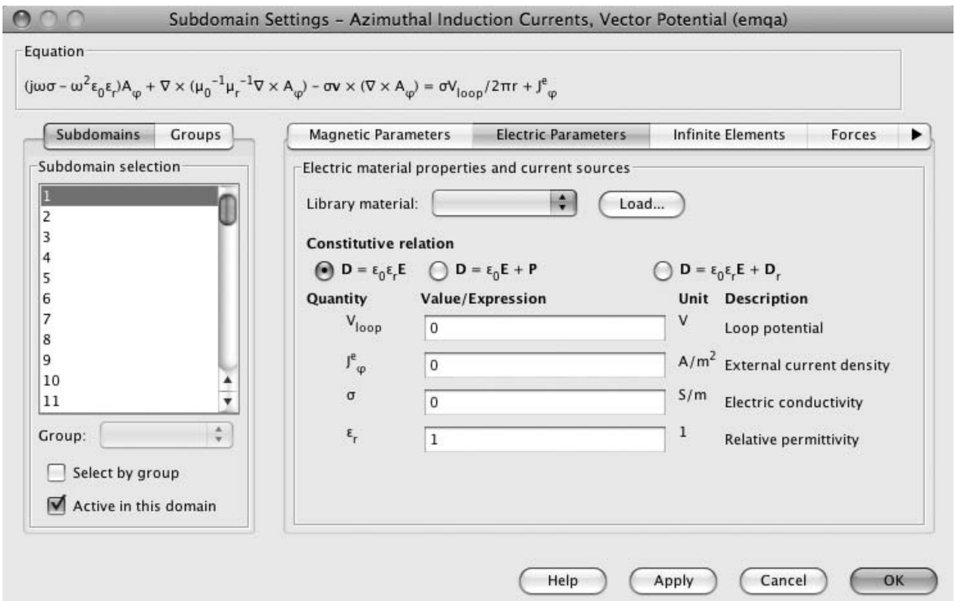


FIGURE 6.169 2D axisymmetric Inductive_Heating_3 model Subdomain Settings (1), Electric Parameters edit window

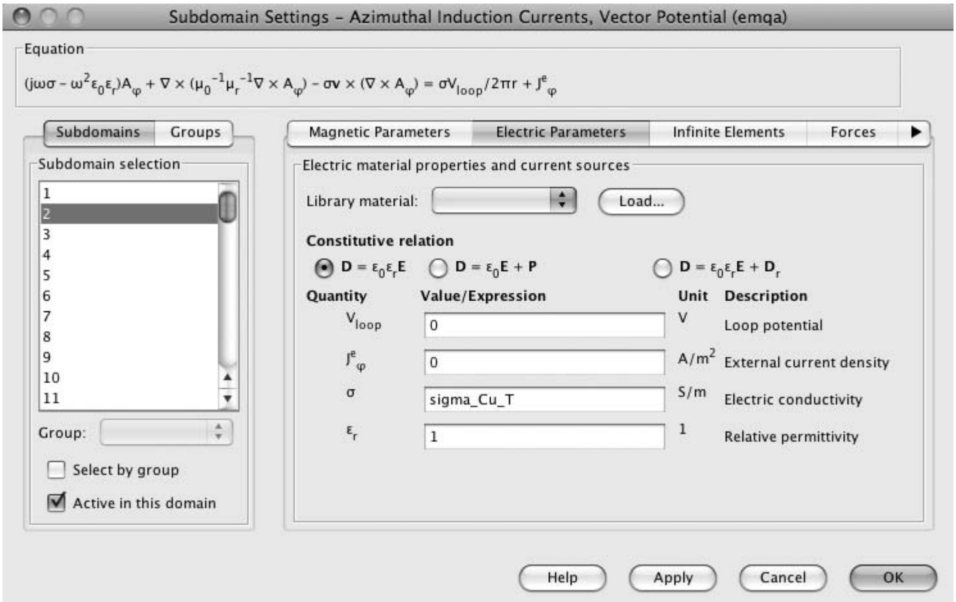


FIGURE 6.170 2D axisymmetric Inductive_Heating_3 model Subdomain Settings (2), Electric Parameters edit window

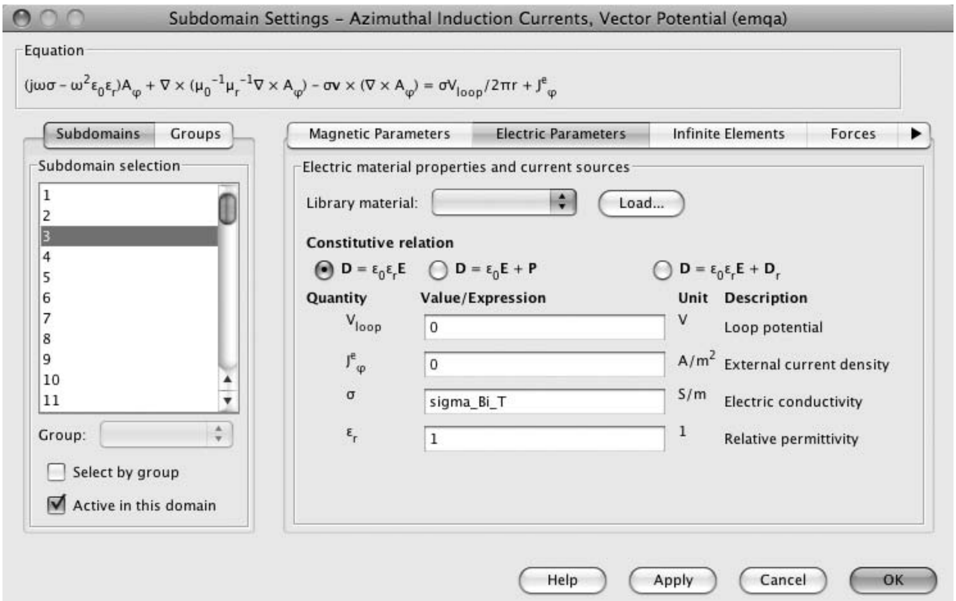


FIGURE 6.171 2D axisymmetric Inductive_Heating_3 model Subdomain Settings (3), Electric Parameters edit window

Table 6.43 Boundary Settings Edit Window

Boundary	Boundary Condition	Figure Number
1, 3–7	Axial symmetry	6.173
2, 9, 14	Magnetic insulation	6.174

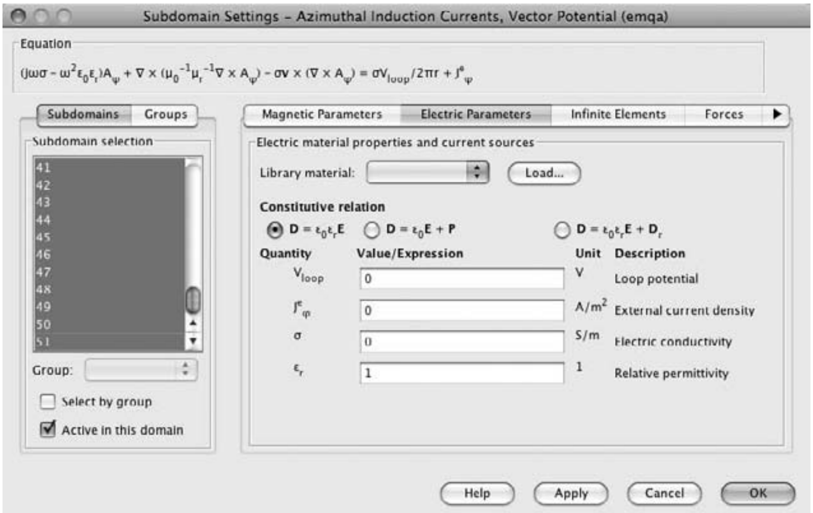


FIGURE 6.172 2D axisymmetric Inductive_Heating_3 model Subdomain Settings (4–51), Electric Parameters edit window

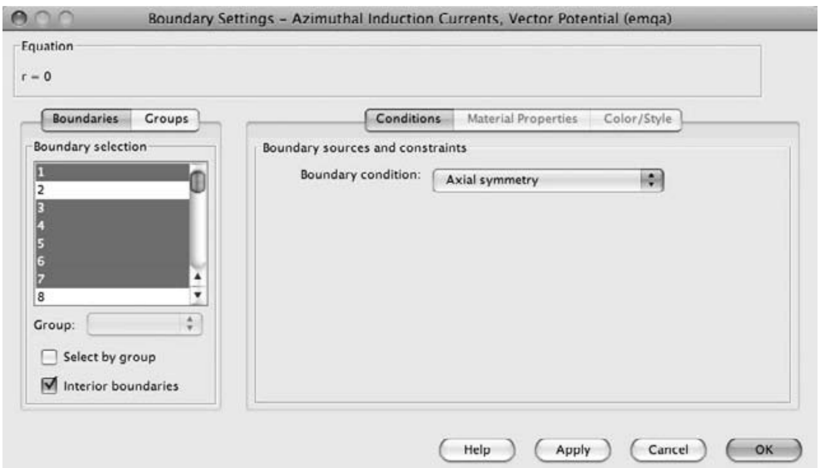


FIGURE 6.173 2D axisymmetric Inductive_Heating_3 model Boundary Settings (1, 3–7) edit window

Table 6.44 Boundary Settings Edit Window

Boundary	Boundary Condition	Value	Figure Number
17–208	Surface current	Js_0	6.175

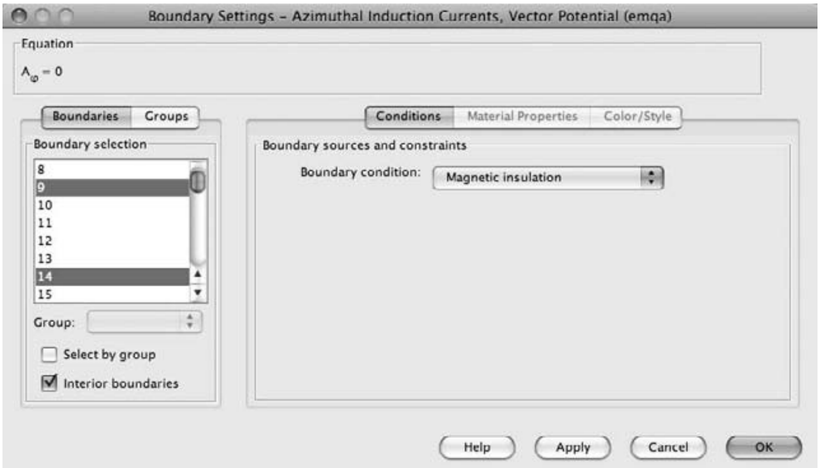


FIGURE 6.174 2D axisymmetric Inductive_Heating_3 model Boundary Settings (2, 9, 14) edit window

In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.44. See Figure 6.175.

Click OK. See Figure 6.176.

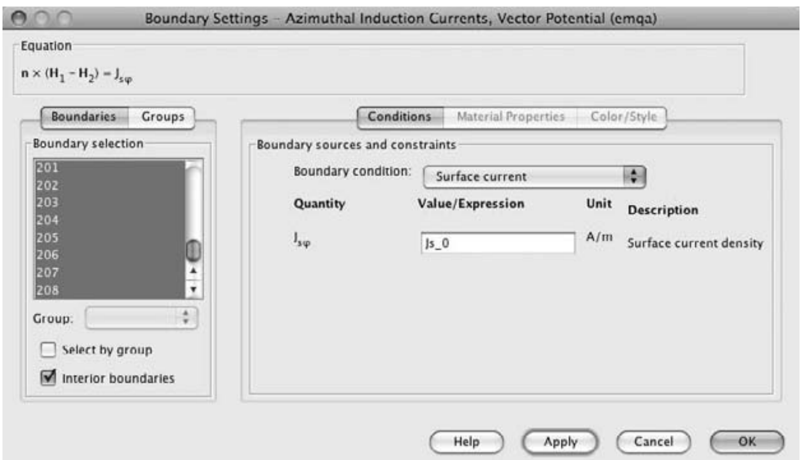


FIGURE 6.175 2D axisymmetric Inductive_Heating_3 model Boundary Settings (17–208) edit window

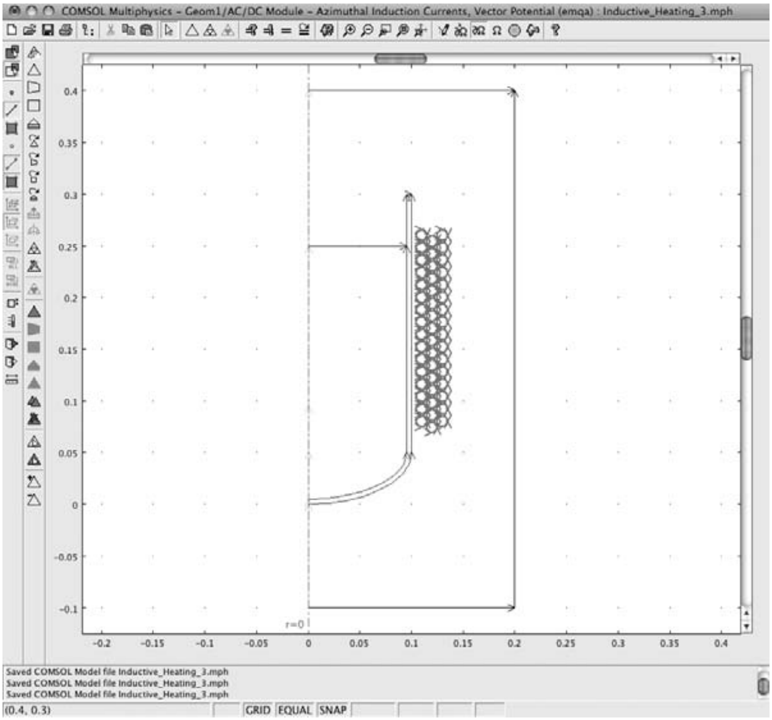


FIGURE 6.176 2D axisymmetric Inductive_Heating_3 model, coil

Physics Subdomain Settings: General Heat Transfer (htgh)

Using the menu bar, select Multiphysics > General Heat Transfer (htgh). Using the menu bar, select Physics > Subdomain Settings > Conduction. In the Subdomain edit windows, enter the information shown in Table 6.45. See Figures 6.177, 6.178, and 6.179.

Table 6.45 Subdomain Edit Window

Subdomain	Name	Expression	Description	Figure Number
1	*k* (isotropic)	k_N2	Thermal conductivity	6.177
	ρ	rho_N2	Density	
	*C*p	Cp_N2	Heat capacity	
2, 4–51	*k* (isotropic)	k_Cu	Thermal conductivity	6.178
	ρ	rho_Cu	Density	
	*C*p	Cp_Cu	Heat capacity	
3	*k* (isotropic)	k_Bi	Thermal conductivity	6.179
	ρ	rho_Bi	Density	
	*C*p	Cp_Bi	Heat capacity	

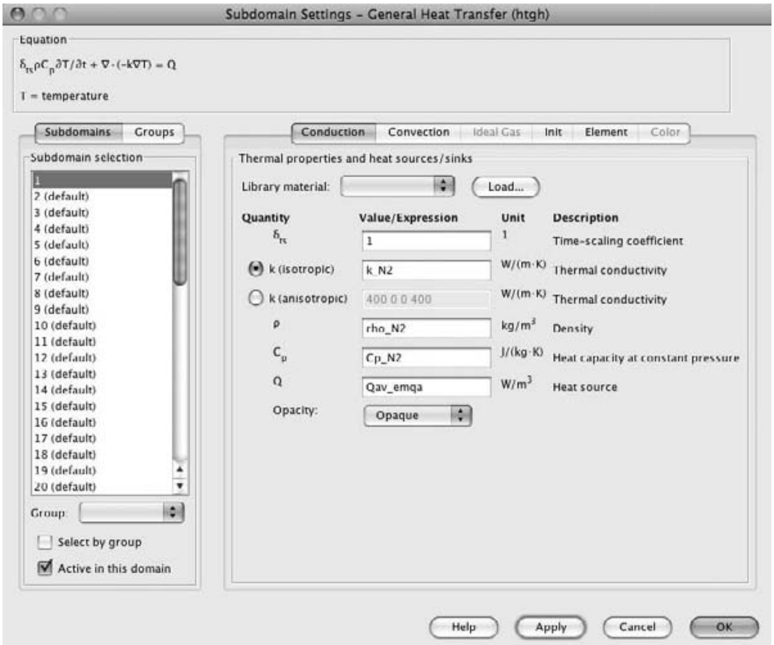


FIGURE 6.177 2D axisymmetric Inductive_Heating_3 model Subdomain Settings (1) edit window

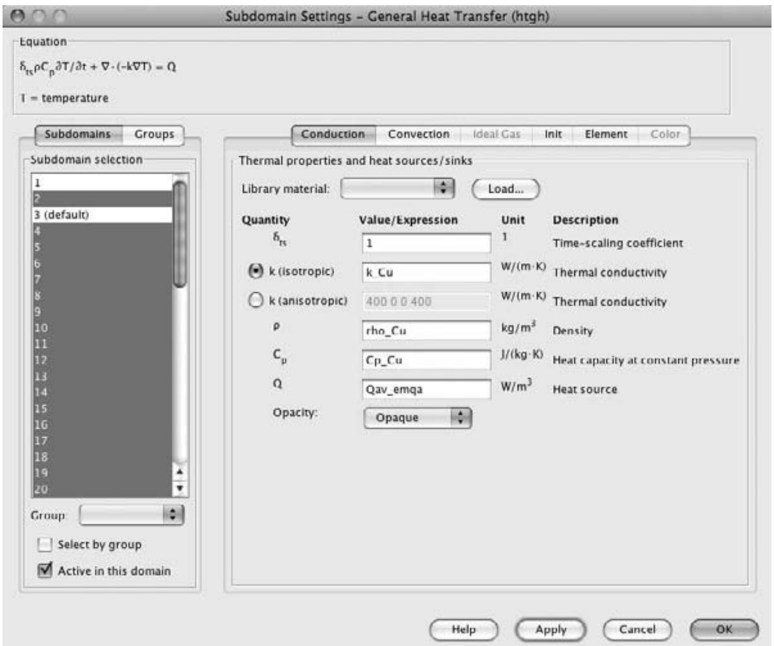


FIGURE 6.178 2D axisymmetric Inductive_Heating_3 model Subdomain Settings (2, 4–51) edit window

FIGURE 6.179 2D axisymmetric Inductive_Heating_3 model Subdomain Settings (3) edit window

Select the Init tab. Select subdomains 1–51. Enter T_refCu in the $T(t_0)$ edit window. See Figure 6.180. Click OK.

Physics Boundary Settings: General Heat Transfer (htgh)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – General Heat Transfer (htgh) edit windows, enter the information shown in Table 6.46. Click OK. See Figures 6.181 and 6.182.

Table 6.46 Boundary Settings Edit Window

Boundary	Boundary Condition	Value	Figure Number
1, 3–7	Axial symmetry	—	6.181
2, 9, 14	Temperature	T_refCu	6.182

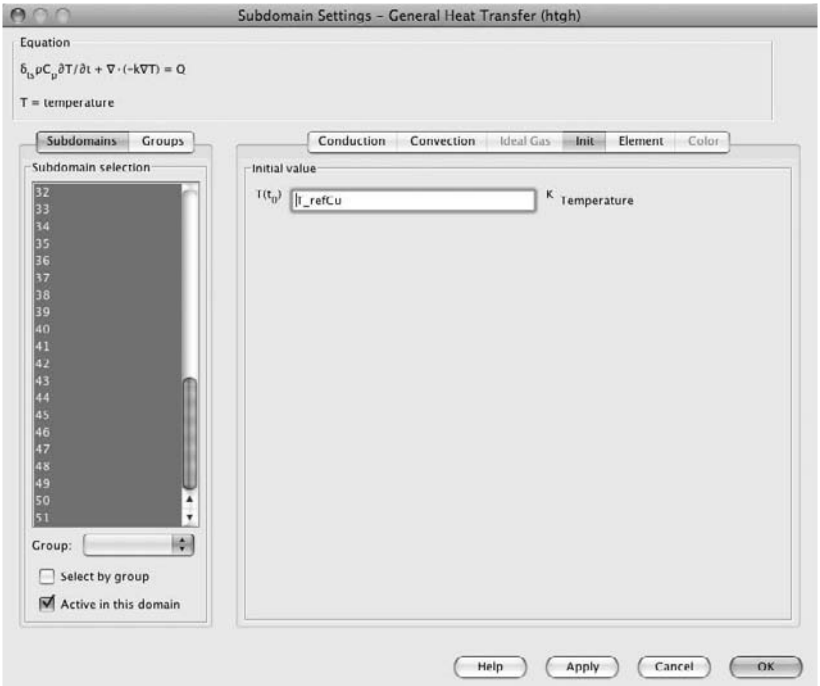


FIGURE 6.180 2D axisymmetric Inductive_Heating_3 model Subdomain Settings (1–51), Init edit window

FIGURE 6.181 2D axisymmetric Inductive_Heating_3 model Boundary Settings (1, 3–7) edit window

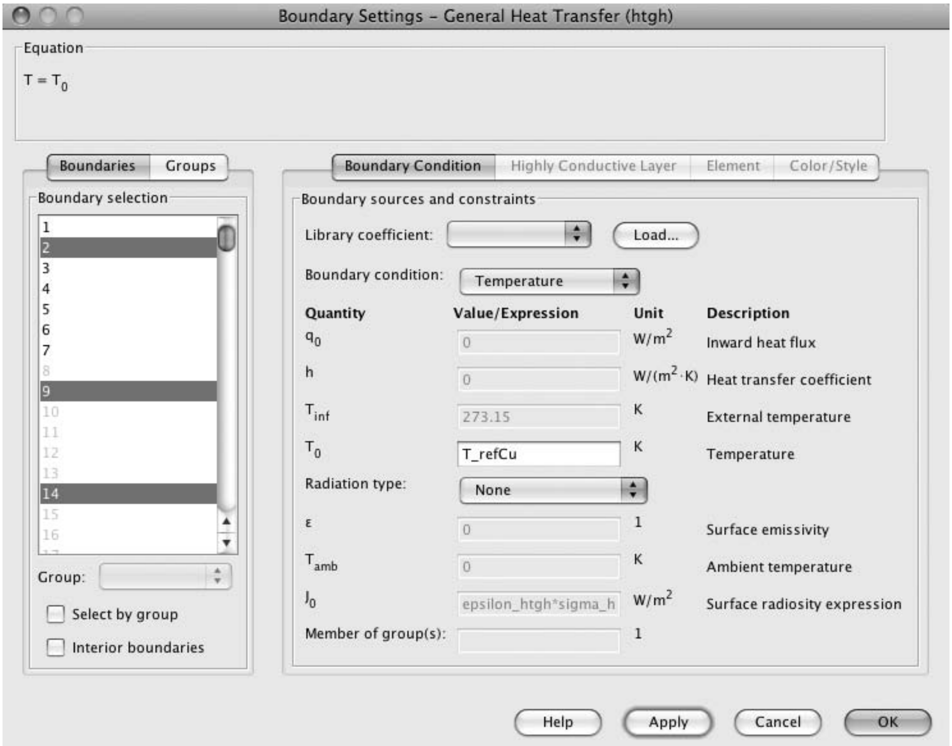


FIGURE 6.182 2D axisymmetric Inductive_Heating_3 model Boundary Settings (2, 9, 14) edit window

Mesh Generation

On the toolbar, click the Initialize Mesh button once. This mesh yields approximately 13,500 elements. See Figure 6.183.

Solving the 2D Axisymmetric Inductive_Heating_3 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Enter linspace(0,1200,21) in the Times edit window. See Figure 6.184.

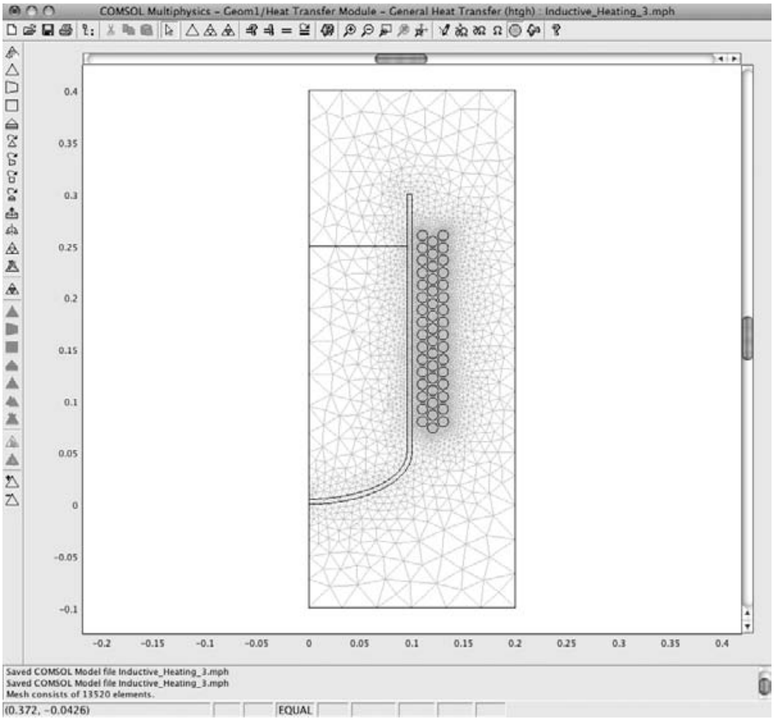


FIGURE 6.183 2D axisymmetric Inductive_Heating_3 model mesh window

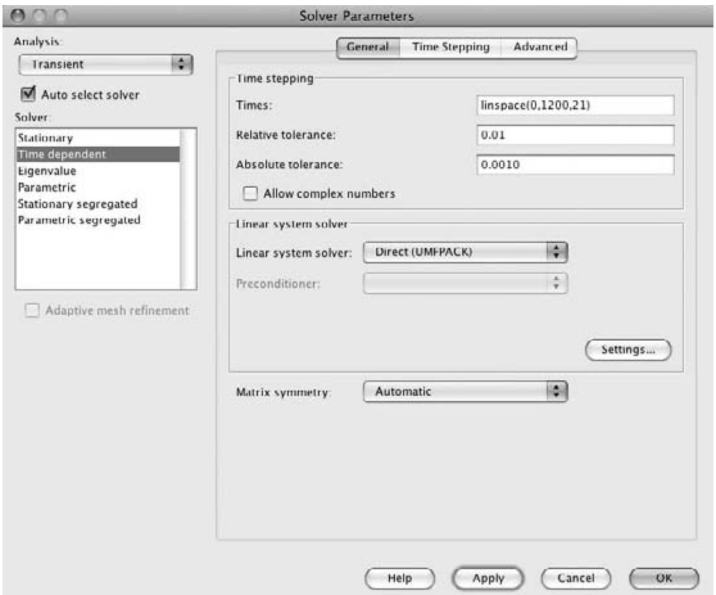


FIGURE 6.184 2D axisymmetric Inductive_Heating_3 model Solver Parameters edit window

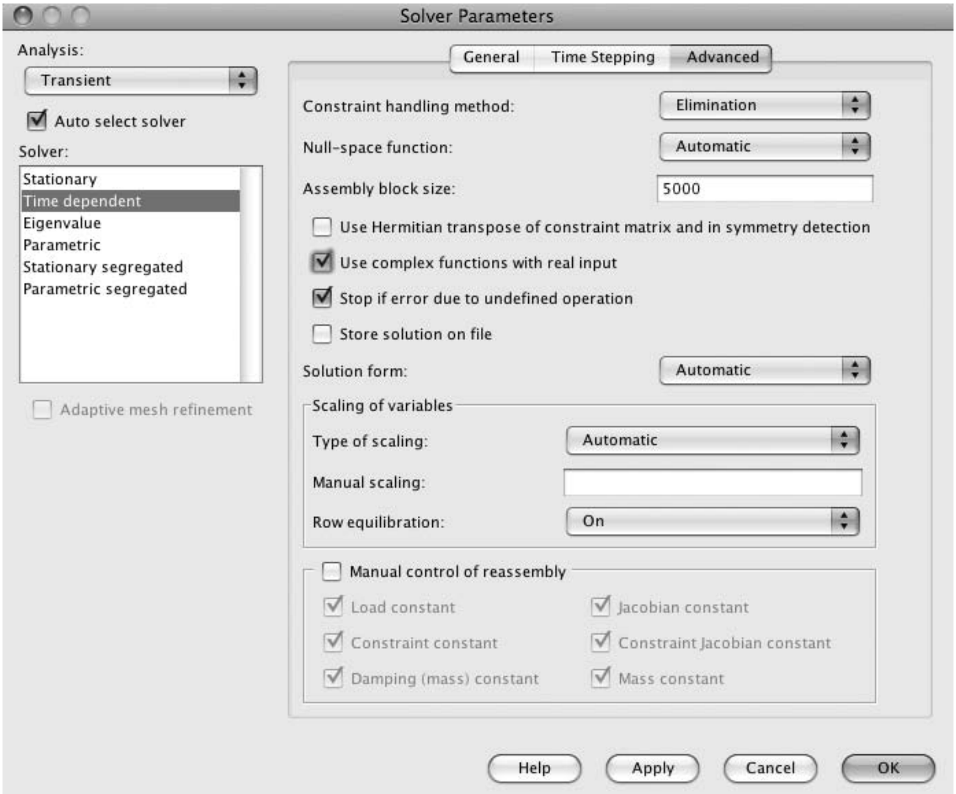


FIGURE 6.185 2D axisymmetric Inductive_Heating_3 model Solver Parameters, Advanced edit window

Click the Advanced button. Check the Use complex functions with real input check box. See Figure 6.185. Click OK.

Time-Dependent Solving of the 2D Resistive_Heating_3 Model

Select Solve > Solve Problem. See Figure 6.186.

Postprocessing and Visualization

The default plot shows the temperature distribution in kelvins. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot Parameters > Surface. Verify that the Surface plot check box is checked and that the Predefined quantities pull-down list shows “Temperature.” Select “degC” or “°C” from the Unit pull-down list. See Figure 6.187.

FIGURE 6.186 2D axisymmetric Inductive_Heating_3 model solution

Click OK. See Figure 6.188.

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.189.

Click the Start Animation button. See Figure 6.190.

Alternatively, you can play the file Movie6_IH_3.avi that was supplied with this book.

2D Axisymmetric Inductive Heating Models: Summary and Conclusions

The models presented in this section of Chapter 6 have introduced the following new concepts: AC induction and mixed-materials modeling. Previously introduced

FIGURE 6.187 2D axisymmetric Inductive_Heating_3 model Plot Parameters edit window

concepts included Ohm's law, Joule heating, mixed-mode modeling, triangular mesh, transient analysis, the good first approximation, and 2D axisymmetric coordinates.

The three 2D axisymmetric inductive heating models demonstrate the difference in level of complexity between single-coil and multi-coil models. In the Inductive_Heating_1 model, the concept of inductively produced heating was introduced. In the Inductive_Heating_2 model, the concept of inductively

FIGURE 6.188 2D axisymmetric Inductive_Heating_3 model degrees Centigrade

produced heating as applied to a practical application (a heated crucible) was used to present one example of the diverse applied scientific and engineering model designs that can be explored using electro-thermal coupling and transient analysis. In the Inductive_Heating_3 model, the crucible was filled with a commonly used metal for melting.

These models are examples of the good first approximation type of model. They demonstrate the significant power of relatively simple physical principles, such as Ohm's law and Joule's law, when applied in the COMSOL Multiphysics modeling environment. They could, of course, be modified by the addition of calculations insulating materials and heat loss through convection, among other changes.

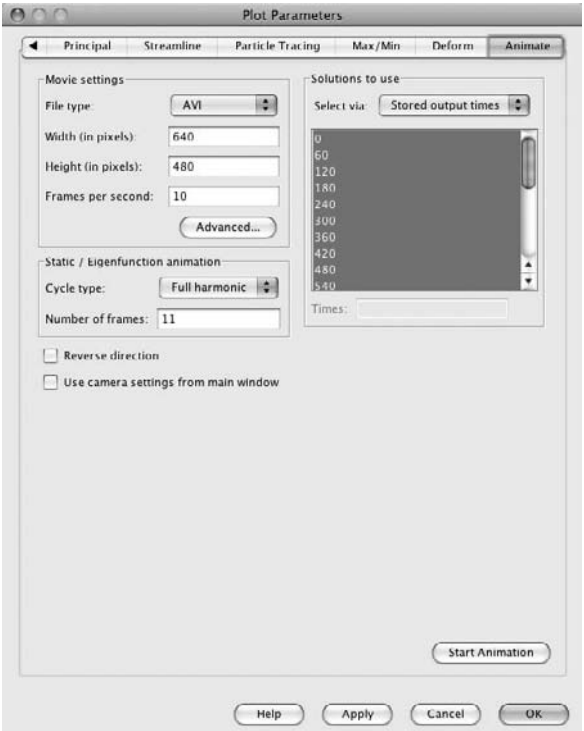


FIGURE 6.189 2D axisymmetric Inductive_Heating_3 model Plot Parameters window

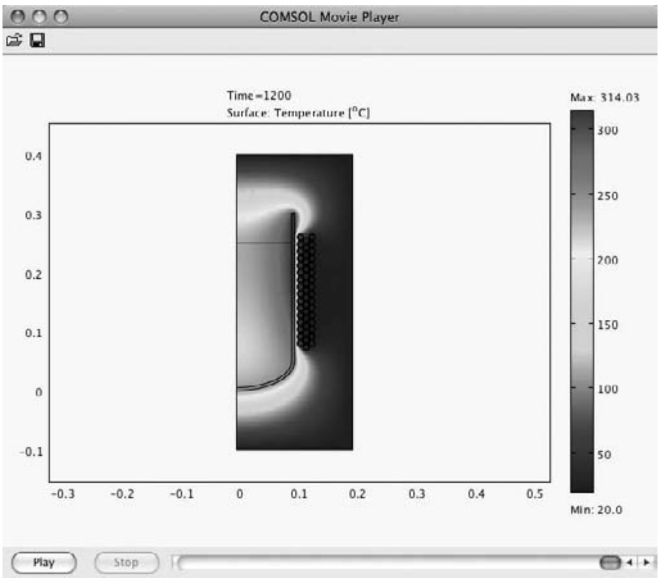


FIGURE 6.190 2D axisymmetric Inductive_Heating_3 model animation, final frame

References

1. http://en.wikipedia.org/wiki/Ohm%27s_Law
2. http://en.wikipedia.org/wiki/Joule_Heating
3. http://en.wikipedia.org/wiki/Direct_current
4. http://en.wikipedia.org/wiki/AC_current
5. http://en.wikipedia.org/wiki/Three_body_problem
6. http://en.wikipedia.org/wiki/Georg_Ohm
7. http://en.wikipedia.org/wiki/James_Joule
8. http://en.wikipedia.org/wiki/Ohm%27s_Law
9. http://en.wikipedia.org/wiki/Joule%27s_Law
10. http://en.wikipedia.org/wiki/Issac_newton
11. http://en.wikipedia.org/wiki/Newton%27s_Law_of_Cooling #Newton.27s_law_of_cooling
12. http://en.wikipedia.org/wiki/Joseph_Fourier
13. http://en.wikipedia.org/wiki/Fourier%27s_Law
14. http://en.wikipedia.org/wiki/Ohm%27s_Law
15. http://en.wikipedia.org/wiki/Joule%27s_Law
16. http://en.wikipedia.org/wiki/James_Joule
17. http://en.wikipedia.org/wiki/Léon_Foucault
18. http://en.wikipedia.org/wiki/Eddy_currents
19. file:///Applications/COMSOL34/doc/acdc/wwhelp/wwhimpl/common/html/wwhelp.htm?context=acdc&file=modeling_acdc.3.4.html#142268
20. http://en.wikipedia.org/wiki/Induction_heating
21. http://www.boulder.nist.gov/div853/lead_free/part2.html#%202.2.4.
22. http://en.wikipedia.org/wiki/Bismuth
23. http://en.wikipedia.org/wiki/Nitrogen

Exercises

1. Build, mesh, and solve the COMSOL 2D resistive heating model problem presented in this chapter.
2. Build, mesh, and solve the first variation of the 2D resistive heating model problem presented in this chapter.
3. Build, mesh, and solve the second variation of the 2D resistive heating model problem presented in this chapter.

4. Build, mesh, and solve the 2D axisymmetric inductive heating model presented in this chapter.
5. Build, mesh, and solve the first variation of the 2D axisymmetric inductive heating model presented in this chapter.
6. Build, mesh, and solve the second variation of the 2D axisymmetric inductive heating model presented in this chapter.
7. Explore other materials as applied in the 2D resistive heating models.
8. Explore other heater geometries similar to those seen in the 2D resistive heating models.
9. Explore how a change of the gas (e.g., $N_2 \rightarrow He$) modifies the behavior of the 2D axisymmetric inductive heating model.
10. Explore how changes in the crucible geometry affect the heating rate of the 2D axisymmetric inductive heating model.

7 2D Complex Mixed-Mode Modeling

In This Chapter

■ 2D Complex Mixed-Mode Guidelines for New COMSOL® Multiphysics® Modelers

2D Complex Mixed-Mode Modeling Considerations

In this chapter, the basic material on 2D modeling presented in Chapters 4, 5, and 6 will be utilized and expanded. In the earlier chapters, models were built and solved using static, quasi-static, and transient methods. In this chapter, all of those methods of solution will be employed. The physics of transient models is intrinsically more difficult

than that for either the static or quasi-static models. Transient models require a firmer understanding of the underlying physical principles being modeled and a more complete (better) characterization of the materials employed in the model.

NOTE In transient or time-dependent (e.g., dynamic, unsteady) models, at least one of the dependent variables changes as a function of time.

The 2D models in this chapter implicitly assume, in compliance with the laws of physics, that the energy flow, the materials properties, the environment, and any other conditions and variables of interest are homogeneous, isotropic, or constant, unless otherwise specified (e.g., time dependent), throughout the entire domain of interest, both within the model and, through the boundary conditions, in the environs of the model.

The three models presented in this chapter—the 2D electric impedance sensor model, the 2D generator model, and the 2D generator sector model—are developed using the AC/DC Module. Each of these three models introduces the modeler to different modeling aspects in the employment of the AC/DC Module to explore a range of similar design, test, and engineering problems.

Electrical resistance tomography is a sensing technology that applies currents, measures the resulting voltages on the surface of a body (e.g., inanimate, animate) and infers impedances from those data. This technology was developed independently in several diverse areas of study (geophysics,[1] industrial process imaging,[2] and medical imaging,[3] to name a few). As a result, substantially the same technology has come to be known by different names in the literature (e.g., electrical resistivity tomography, electrical resistance tomography, electrical impedance tomography). This technology is widely used in the previously mentioned areas and is one of the most promising noninvasive measurement techniques available.

The 2D electric impedance sensor models, both basic and advanced, employ the high-frequency currents (1 MHz alternating currents AC[4]). These currents are applied to the material of the modeled body to locate volumes that differ in impedance from the impedance of the bulk material by monitoring the local impedance. The basic 2D electric impedance sensor model detects the location of a fixed volume difference area. The advanced 2D electric impedance sensor model detects the location of a fluctuating difference volume, such as might be seen in a medical application measuring lung function. 2D electric impedance tomography research is exploring the application of this measurement technology to the detection of breast cancer,[5] lung function,[6] brain function,[7] and numerous other areas.

The 2D generator model introduces the modeler to rotary motion and the conversion of mechanical energy to electrical energy.[8] The 2D generator sector model employs symmetry to reduce the geometric difficulty of building the generator model and adds an ordinary differential equation[9] to handle the mechanical dynamics and calculate the torque caused by magnetic forces.

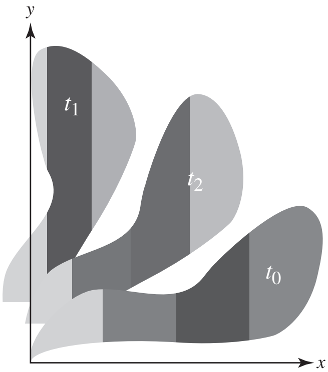


FIGURE 7.1 2D coordinate system, plus time

2D Coordinate System

In a steady-state solution to a 2D model, parameters can vary only as a function of position in space (x) and space (y) coordinates. Such a 2D model represents the parametric condition of the model in a time-independent mode (quasi-static). In a transient solution model, parameters can vary both by position in space (x), and space (y) and in time (t); see Figure 7.1.

The transient solution model is essentially a sequential collection of (quasi-static) solutions, except that one or more of the dependent variables [$f(x, y, t)$] has changed with time. The space coordinates (x) and (y) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values ($x_{min} <= x <= x_{max}$) and ($y_{min} <= y <= y_{max}$). The time coordinate (t) represents the range of values ($t_{min} <= t <= t_{max}$) from the beginning of the observation period (t_{min}) to the end of the observation period (t_{max}).

Electrical Impedance Theory

The concept of electrical impedance,[10] as used in alternating current (AC) theory, is an expansion of the basic concept of resistance as exemplified by Ohm's law,[11] in direct current theory.

NOTE Ohm's law was discovered by Georg Ohm and published in 1827:

$$I = \frac{V}{R} \tag{7.1}$$

where I = current in amperes (A)
V = voltage (electromotive force) in volts (V)
R = resistance in ohms

In AC theory, both voltage (V) and current (I) alternate periodically as a function of time. Typically, the alternating behavior—frequency (f)—of the voltage and current are separately represented either as a single sinusoidal wave or as a sum of several sinusoidal waves.

NOTE The analysis of complex waveforms is handled by Fourier analysis.[12] That topic will not be presented here. However, modelers are encouraged to expand their technological horizons by exploring the subject of waveform analysis further.

In this case, for clarity, the exploration of the concept of impedance will be confined to single frequency analysis. The concept of impedance was developed and named by Oliver Heaviside[12] in 1886. Impedance was reformulated in the currently used complex number formulation by Arthur E. Kennelley[13] in 1893.

The first factor that needs to be considered, when expanding modeling calculations from the DC realm [frequency equals zero ($f = 0$)] to the AC realm [frequency greater than zero ($f > 0$)], is that the resistance (R) maps into the impedance (Z) as follows:[14]

$$Z = R + j\left(\omega L - \frac{1}{\omega C}\right) = R + jX = \left(R^2 + X^2\right)^{1/2} e^{j\tan^{-1}(X/R)} \tag{7.2}$$

where
Z = complex impedance (A)
R = resistance (ohm)
$j = (-1)^{1/2}$
$\omega = 2\pi f$ = angular frequency[15]
X = reactance (ohm)[16]
L = inductance (henry)
C = capacitance (farad)

The relative vector-phase relationship of an AC voltage applied to a simple series circuit containing resistance, inductance, and capacitance is shown in Figure 7.2.

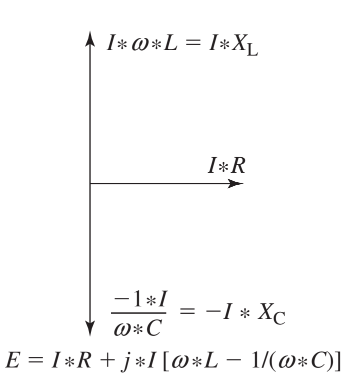


FIGURE 7.2 AC voltage resistive/reactive vector phase diagram

A second factor that needs to be considered by the modeler, when modeling in the AC realm, is the skin depth (δ).[17] In any material, as a function of the complex permittivity,[18] electromagnetic waves (AC) will be attenuated (i.e., dissipated, turned into heat) and shifted in phase as a function of the distance (depth) traveled in that material. As an example, for a transverse electromagnetic wave propagating in the z direction, the voltage relationship would be expressed as follows:

$$E_x = E_0 {*} e^{-kz} = E_0 {*} e^{-\alpha z} {*} e^{-j {*} \beta z} \tag{7.3}$$

where E_x = transverse electromagnetic propagating in the z direction
E_0 = scalar voltage amplitude
k = complex propagation constant
$j = (-1)^{1/2}$
e = base of natural logarithms
α = attenuation constant
β = wave solution constant

where α is

$$\alpha = \omega {*} \left(\frac{\mu\varepsilon}{2}\left(1 + \left(1 + \left(\frac{\sigma}{\omega\varepsilon}\right)^2\right)^{\frac{1}{2}}\right)\right)^{\frac{1}{2}}$$

and where

ε = permittivity
μ = permeability
ω = angular frequency
σ = conductivity

For a good conductor, where $1 << \sigma/\omega\varepsilon$, the 1's in the preceding equation can be ignored and α becomes

$$\alpha = \sqrt{\frac{\omega\mu\sigma}{2}} \tag{7.4}$$

The skin depth (δ) is the point at which the amplitude decreases to $E_0{*}\ e^{-1}$ and, therefore, is

$$\delta = \frac{1}{\alpha} \tag{7.5}$$

The first example presented in this chapter, the basic 2D electric impedance sensor model (2D_EIS_1 model), explores the sensing of multiple small-volume differential conductivity regions. The model is solved for a material that has a bulk conductivity of $1\ e^{-3}$ S/m. The model is implemented using the AC/DC Module Small In-Plane Currents Application Mode and solved using a Stationary solver.

In the advanced 2D electric impedance sensor model (2D_EIS_2 model), a new quasi-static model is built to explore a configurational change using the AC/DC Module Small In-Plane Currents Application Mode and solved using a Parametric solver.

2D Electric Impedance Sensor Model: Basic

The following numerical solution model (2D_EIS_1) is derived from a model that was originally developed by COMSOL® as a Multiphysics® General Industrial Applications demonstration model. That model was developed for distribution with the Multiphysics software as a COMSOL Multiphysics model in the AC/DC Module Model Library.

NOTE As mentioned earlier in this chapter, knowing the skin depth (δ) model limitations is important. For this model, the parameters are as follows:

$$\omega = 2\pi f = 2*3.14159*1E6 = 9.9892E7 \tag{7.6}$$

$$\mu = 4\pi E - 7\,[H/m] = 1.2566E - 6\,[H/m] \tag{7.7}$$

$$\sigma = 1E - 3\,[S/m] \tag{7.8}$$

$$\begin{aligned}\delta = \left(\frac{2}{\omega\mu\sigma}\right)^{\frac{1}{2}} &= \left(\frac{2}{9.9892E7*1.2566E - 6*1E - 3}\right) \\ &= 15.933\,[m]\end{aligned} \tag{7.9}$$

Because the largest dimensions in the model are approximately 1 meter (m), the skin depth (δ) consideration will pose no problem and will not have to be factored into the calculation. (This is a first principles observation.)

To start building the 2D_EIS_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from the Space dimension pull-down list. Select AC/DC Module > Quasi-Statics, Electric > In-Plane Electric Currents > Time-harmonic analysis. See Figure 7.3. Click OK.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 7.1; see also Figure 7.4. Click OK.

NOTE When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar

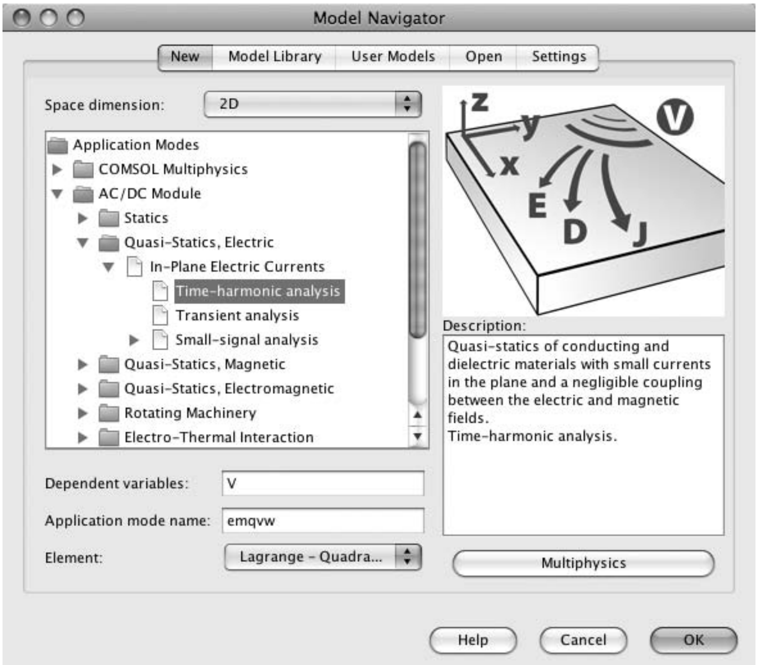


FIGURE 7.3 2D_EIS_1 Model Navigator setup

Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Table 7.1 Constants Edit Window

Name	Expression	Description
sig_bulk	1[mS/m]	Bulk conductivity
eps_r_bulk	5	Relative permittivity in bulk
x_0	−0.35[m]	x position of cavity center
y_0	−0.15[m]	y position of cavity center
r_0	0.09[m]	Cavity radius
x_1	0.0[m]	x position of cavity center
y_1	−0.3[m]	y position of cavity center
r_1	0.12[m]	Cavity radius
x_2	0.35[m]	x position of cavity center
y_2	−0.15[m]	y position of cavity center
r_2	0.06[m]	Cavity radius

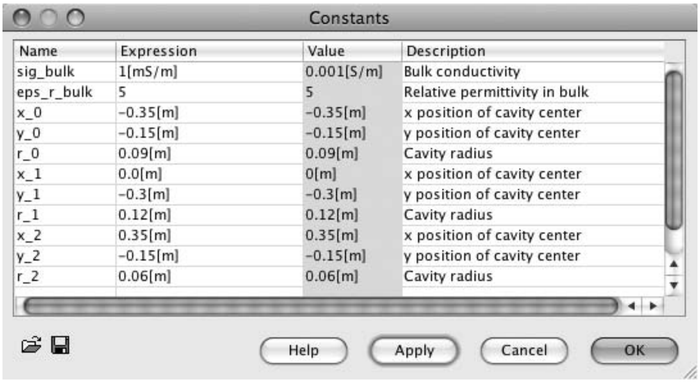


FIGURE 7.4 2D_EIS_1 model Constants edit window

Select File > Save As. Enter 2D_EIS_1 in the Save As edit window. See Figure 7.5. Click the Save button.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 1.0 and a height of 0.5. Select "Base: Corner" and set x equal to −0.5 and y equal to −0.5 in the Rectangle edit window. See Figure 7.6.

Click OK, and then click the Zoom Extents button. See Figure 7.7.

Using the menu bar, select Draw > Specify Objects > Point. In the Point edit window, enter x: −0.01 space 0.01, y: 0 space 0. See Figure 7.8.

Click OK. See Figure 7.9.

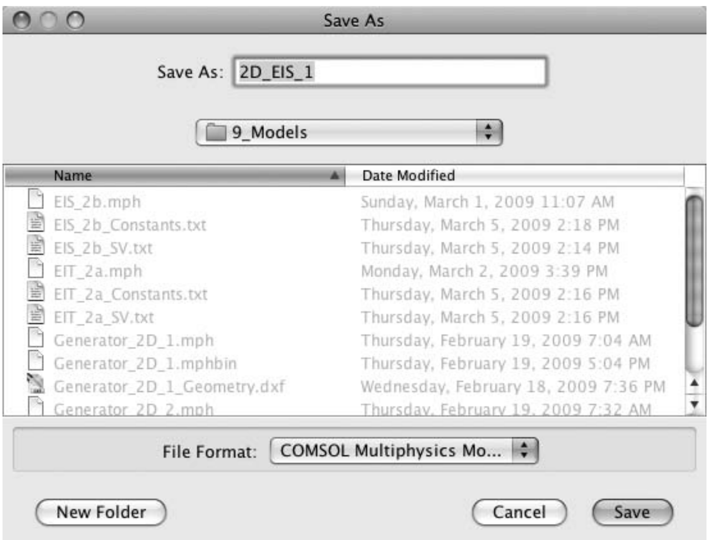


FIGURE 7.5 2D_EIS_1 model Save As edit window

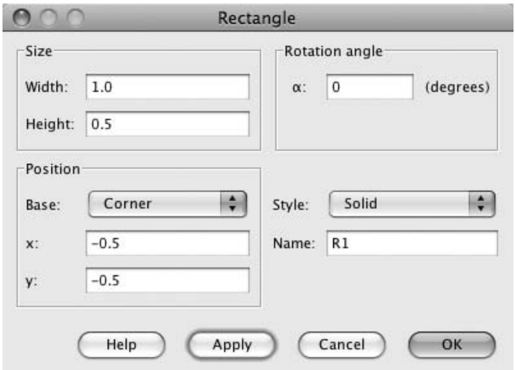


FIGURE 7.6 2D_EIS_1 model Rectangle edit window

NOTE The rectangle is the 2D representation of a 3D rectangular body in cross section. The points are added to the 2D rectangle to define the location of the electrode (between the points) on the boundary of the rectangle.

FIGURE 7.7 2D_EIS_1 model rectangle

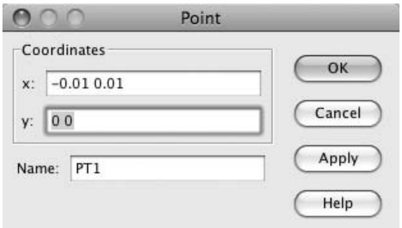


FIGURE 7.8 2D_EIS_1 model Point edit window

Physics Settings: Scalar Expressions

Using the menu bar, select Options > Expressions > Scalar Expressions. In the Scalar Expressions edit window, enter the information shown in Table 7.2; see Figure 7.10. Click OK.

Physics Settings: Scalar Variables

Select Physics > Scalar Variables. Enter 1e6 in the nu_emqvw Application Scalar Variables edit window. See Figure 7.11. Click OK.

FIGURE 7.9 2D_EIS_1 model rectangle and points

Table 7.2 Scalar Expressions Edit Window

Name	Expression	Description
sigma_0	sig_bulk*(((x−x_0)^2+(y−y_0)^2)>r_0^2)	Conductivity bulk
epsilon_r_0	1+(eps_r_bulk-1)*(((x−x_0)^2+(y−y_0)^2)>r_0^2)	Permittivity relative
sigma_1	sig_bulk*(((x−x_1)^2+(y−y_1)^2)>r_1^2)	Conductivity bulk
epsilon_r_1	1+(eps_r_bulk-1)*(((x−x_1)^2+(y−y_1)^2)>r_2^2)	Permittivity relative
sigma_2	sig_bulk*(((x−x_2)^2+(y−y_2)^2)>r_1^2)	Conductivity bulk
epsilon_r_2	1+(eps_r_bulk-1)*(((x−x_2)^2+(y−y_2)^2)>r_2^2)	Permittivity relative
sigma_tot	(sigma_0+sigma_1+sigma_2)/3	Conductivity total
epsilon_r_tot	(epsilon_r_0+epsilon_r_1+epsilon_r_2)/3	Permittivity total

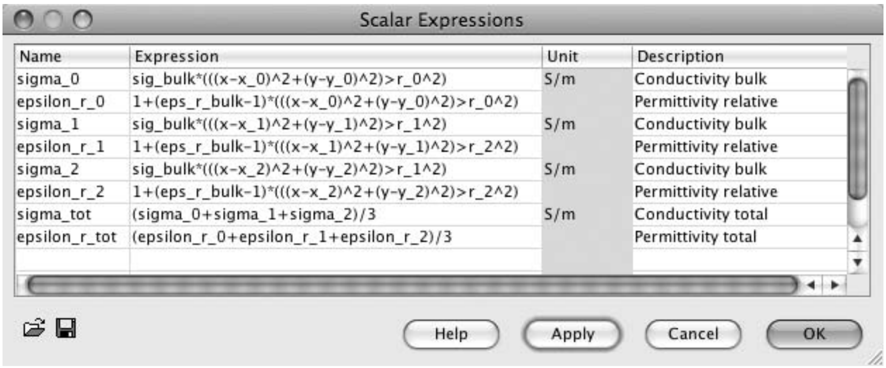


FIGURE 7.10 2D_EIS_1 model Scalar Expressions edit window

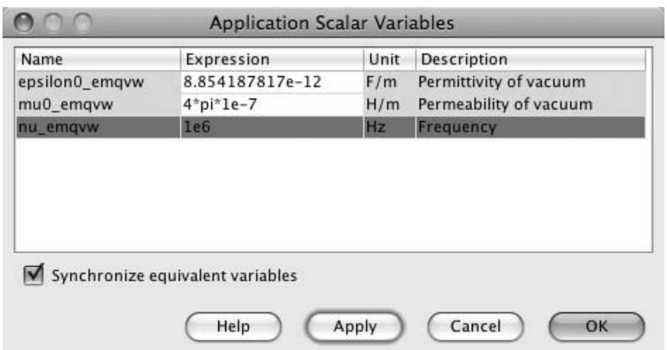


FIGURE 7.11 2D_EIS_1 model Application Scalar Variables edit window

Table 7.3 Subdomain Edit Windows

Name	Expression	Description
σ (isotropic)	sigma_tot	Electrical conductivity
εr (isotropic)	epsilon_r_tot	Relative permittivity

Physics Subdomain Settings: In-Plane Electric Currents (emqvw)

Having established the geometry for the 2D_EIS_1 model of a block with an electrode, the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). In the Subdomain edit windows, enter the information shown in Table 7.3.

Click the $D = \varepsilon_0\varepsilon_r E$ radio button. See Figure 7.12. Click OK.

Physics Boundary Settings: In-Plane Electric Currents (emqvw)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 7.4. See Figures 7.13, 7.14, and 7.15.

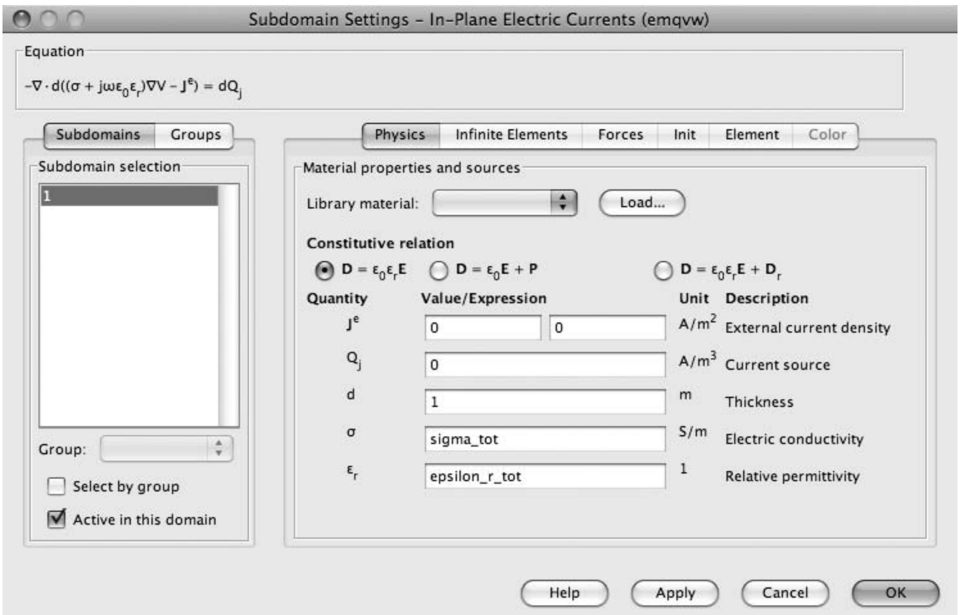


FIGURE 7.12 2D_EIS_1 model Subdomain Settings edit window

Table 7.4 Boundary Settings – In-Plane Electric Currents (emqvw) Edit Window

Boundary	Boundary Condition	Figure Number
1, 2, 6	Ground	7.13
3, 5	Electrical insulation	7.14
4	Port	7.15

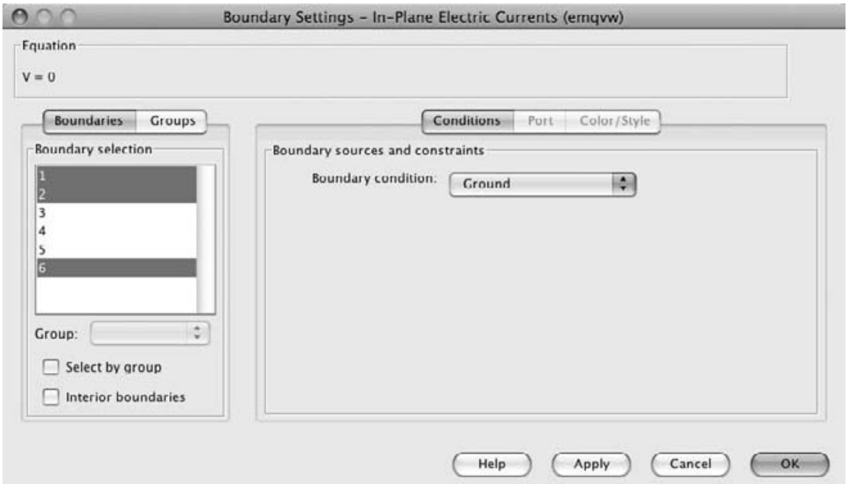


FIGURE 7.13 2D_EIS_1 model Boundary Settings (1, 2, 6) edit window

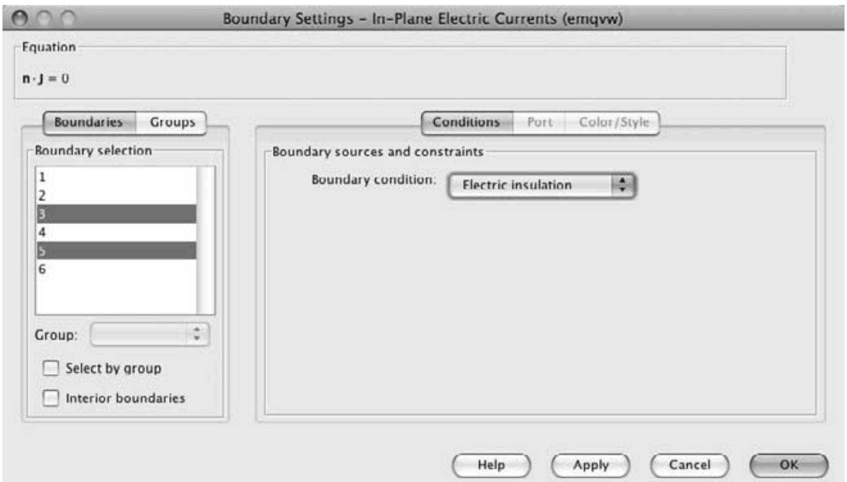


FIGURE 7.14 2D_EIS_1 model Boundary Settings (3, 5) edit window

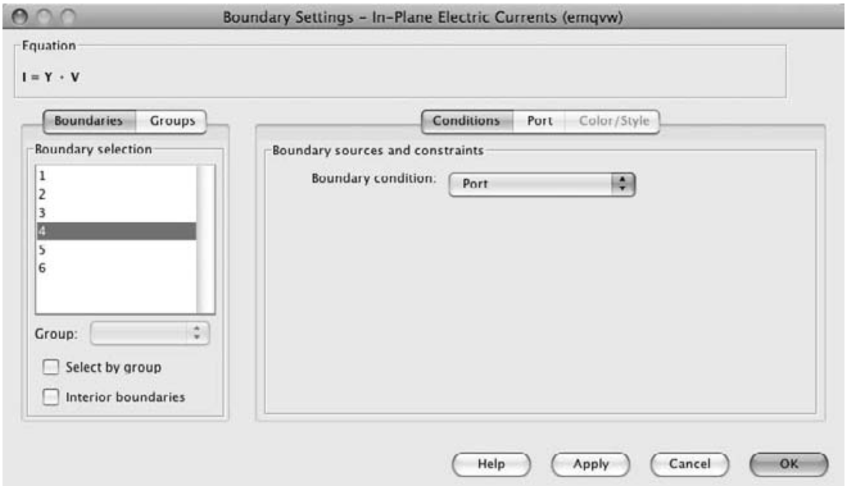


FIGURE 7.15 2D_EIS_1 model Boundary Settings (4) edit window

Select the Port tab. Check Use port as input. Select "Fixed current density" from the input property pull-down list. See Figure 7.16. Click OK.

Mesh Generation

From the toolbar, select Mesh > Free Mesh Parameters > Global. Select Predefined mesh sizes > Normal (from the pull-down list). Select "Custom mesh size." Enter 0.01 in the Maximum element size edit window. See Figure 7.17.

Click the Remesh button, and then click OK. See Figure 7.18.

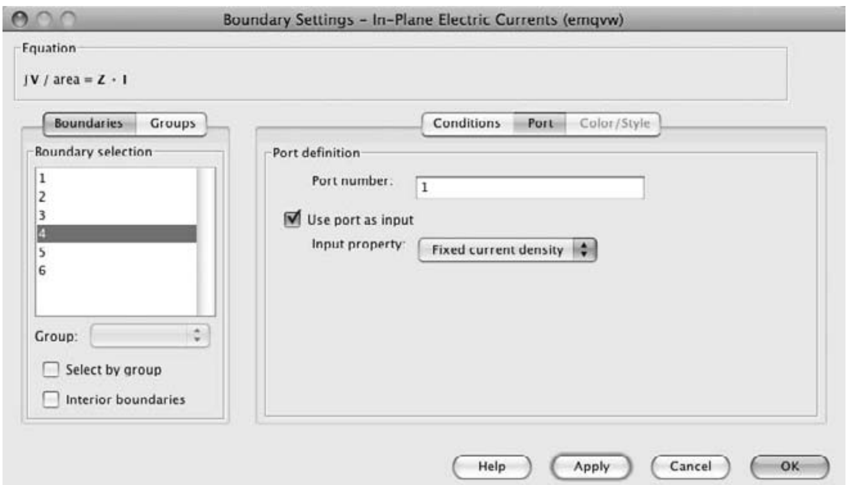


FIGURE 7.16 2D_EIS_1 model Boundary Settings (Port) edit window

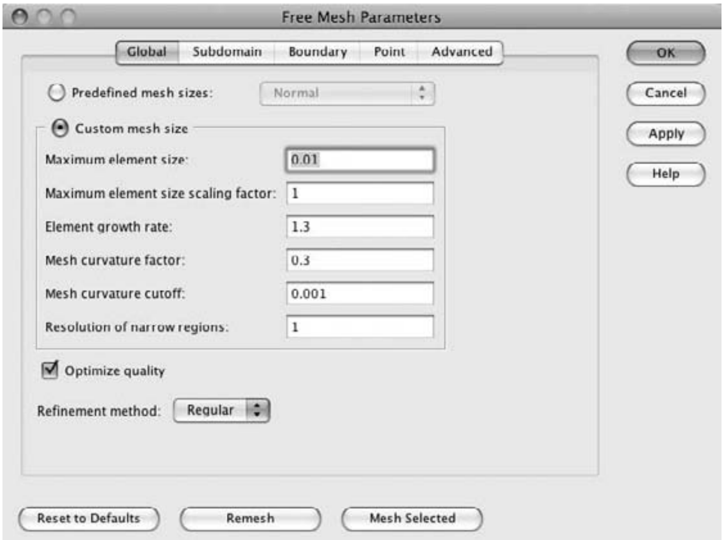


FIGURE 7.17 2D_EIS_1 model Free Mesh Parameters edit window

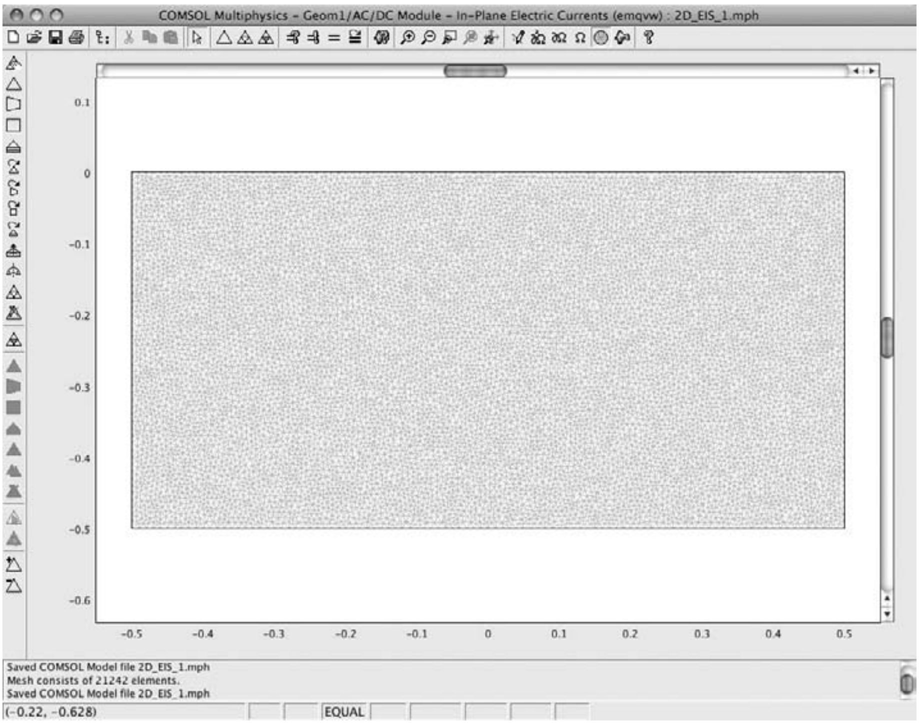


FIGURE 7.18 2D_EIS_1 model mesh

FIGURE 7.19 2D_EIS_1 model Solver Parameters edit window

Solving the 2D_EIS_1 Model

Using the menu bar, select Solve > Solver Parameters.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Select "Stationary Solver." See Figure 7.19. Click OK.

Using the menu bar, select Solve > Solve Problem. See Figure 7.20.

Postprocessing and Visualization

The default plot shows a surface plot of the electric potential (V) distribution in volts. To visualize the detected regions of differential conductivity, the plot parameters will need to be modified.

FIGURE 7.20 2D_EIS_1 model solution

Select Postprocessing > Plot Parameters > Surface. Select "Total current density, norm" from the Predefined quantities pull-down list. Change the expression in the edit window from normJ_emqvw to 20*log10(normJ_emqvw). Click the Range button. Unselect the Auto check box. Enter −35 in the Min edit window and 35 in the Max edit window; see Figure 7.21. Click OK.

Select "wave" from the Colormap pull-down list. See Figure 7.22.

Click OK. See Figure 7.23.

Basic 2D Electric Impedance Sensor Model: Summary and Conclusions

The basic 2D electric impedance sensor model has been built and operated. This model employs a high-frequency current (1 MHz alternating current AC) to explore the differential impedance within the body of a material in a noninvasive manner. Such 1 MHz currents are applied to the material of the modeled body to locate volumes that differ in impedance from the impedance of the bulk material by monitoring the local impedance, as shown in Figure 7.23. The basic model shows the location of three areas of fixed-volume impedance difference.

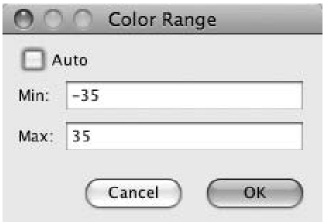


FIGURE 7.21 2D_EIS_1 model solution, Color Range edit window

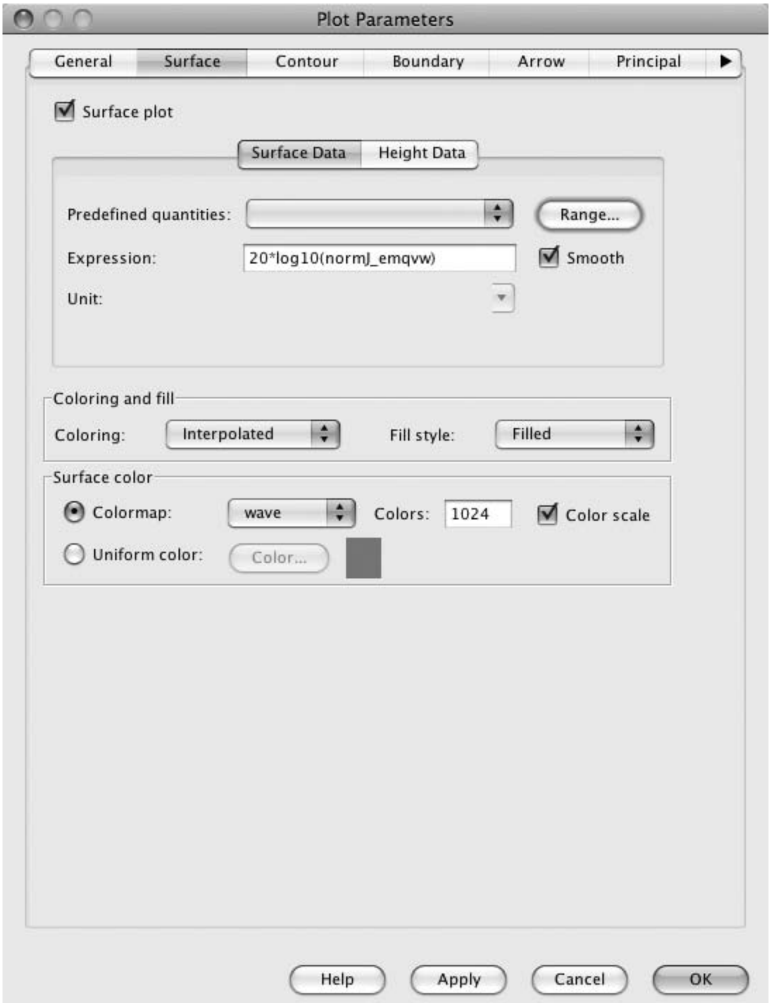


FIGURE 7.22 2D_EIS_1 model solution Plot Parameters edit window

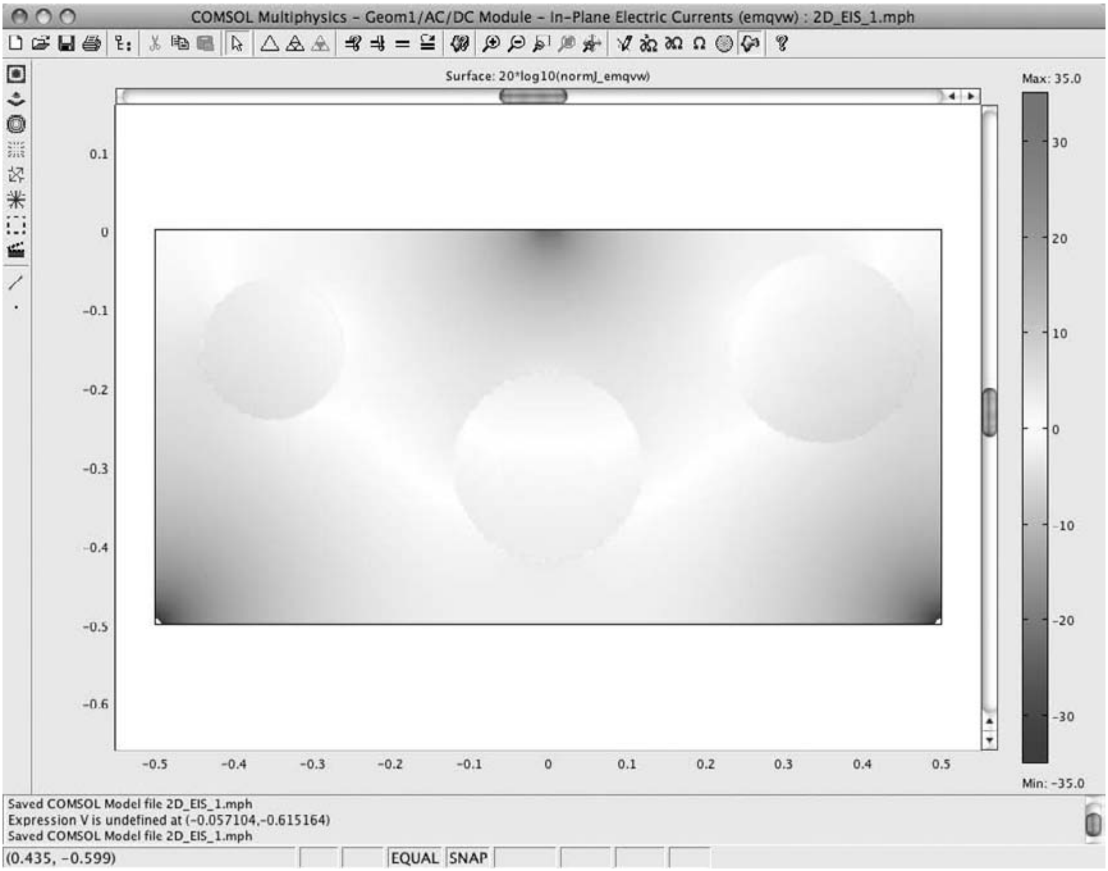


FIGURE 7.23 2D_EIS_1 model solution with detected areas

2D Electric Impedance Sensor Model: Advanced

To start building the 2D_EIS_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from the Space dimension pull-down list. Select AC/DC Module > Quasi-Statics, Electric > In-Plane Electric Currents > Time-harmonic analysis. See Figure 7.24. Click OK.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 7.5; see also Figure 7.25. Click OK.

Select File > Save As. Enter 2D_EIS_2 in the Save As edit window. See Figure 7.26. Click the Save button.

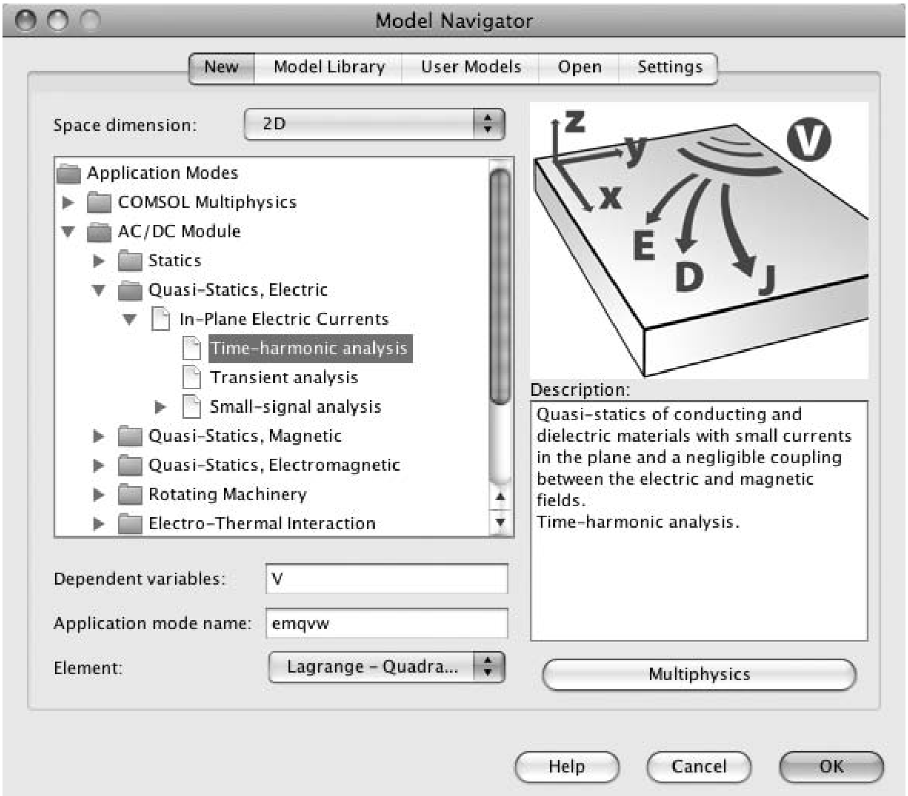


FIGURE 7.24 2D_EIS_2 Model Navigator setup

Table 7.5 Constants Edit Window

Name	Expression	Description
sig_bulk	1[mS/m]	Bulk conductivity
eps_r_bulk	5	Relative permittivity in bulk
x_0	−0.12[m]	x position of cavity center
y_0	0[m]	y position of cavity center
r_0	0.07[m]	Cavity radius
x_1	0.12[m]	x position of cavity center
y_1	0[m]	y position of cavity center
r_1	0.07[m]	Cavity radius
t_0	0	Time init

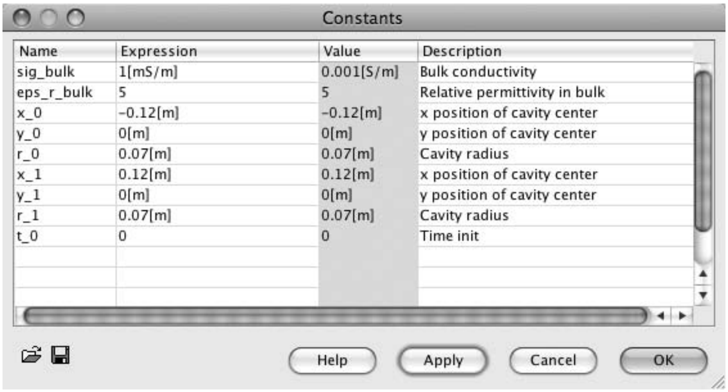


FIGURE 7.25 2D_EIS_2 model Constants edit window

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes of 0.3 and B-semiaxes of 0.15. Select "Base: Center" and set x equal to 0 and y equal to 0 in the Ellipse edit window. See Figure 7.27.

Click OK, and then click the Zoom Extents button. See Figure 7.28.

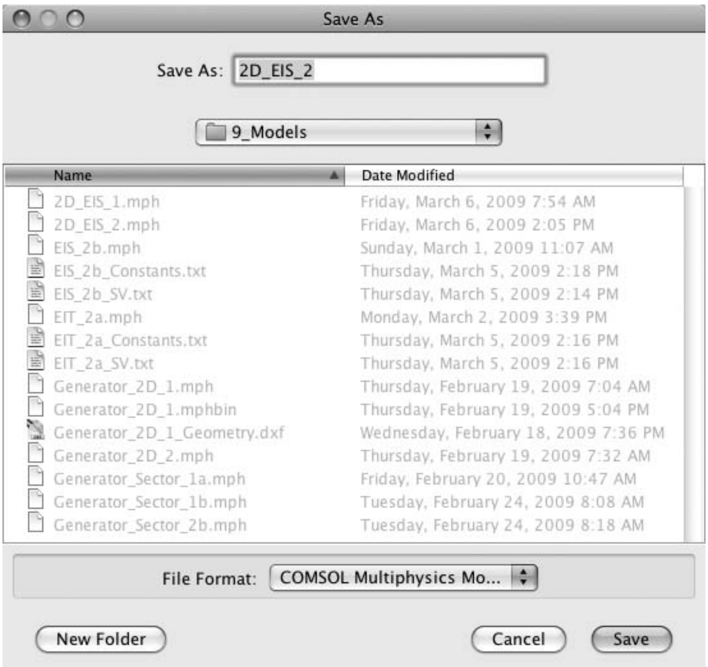


FIGURE 7.26 2D_EIS_2 model Save As edit window

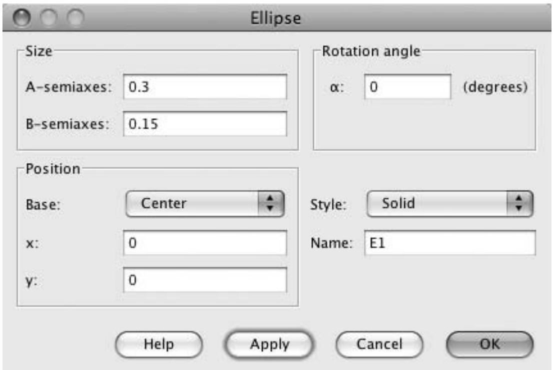


FIGURE 7.27 2D_EIS_2 model Ellipse edit window

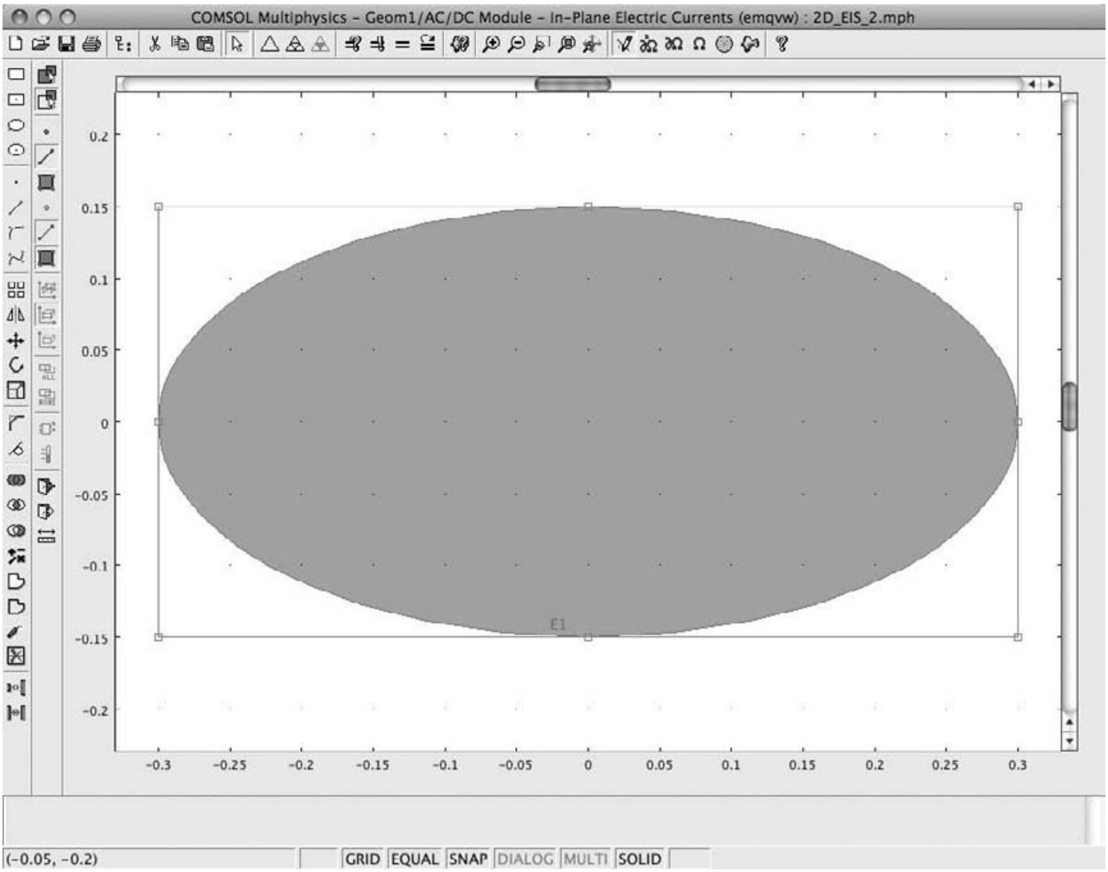


FIGURE 7.28 2D_EIS_2 model rectangle

FIGURE 7.29 2D_EIS_2 model Rectangle edit window

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.04 and a height of 0.001. Select “Base: Corner” and set x equal to −0.02 and y set equal to 0.15–0.001 in the Rectangle edit window. See Figure 7.29. Click OK.

NOTE The purpose of adding the rectangle to the ellipse is to provide a known placement location for the points that will define the edges of the electrode.

Using the menu bar, select Draw > Create Composite Object. Enter E1+R1 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 7.30.

Click OK. See Figure 7.31.

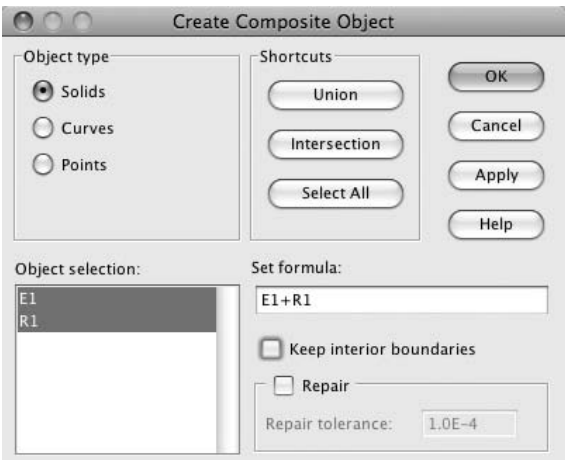


FIGURE 7.30 2D_EIS_2 model Create Composite Object edit window

FIGURE 7.31 2D_EIS_2 model Create Composite Object result

Using the menu bar, select Draw > Specify Objects > Point. In the Point edit window, enter −0.02 0.02 for x and 0.15 0.15 for y. See Figure 7.32.

Click OK. See Figure 7.33.

NOTE The ellipse is the 2D representation of a 3D elliptical body in cross section (e.g., similar to the cross section that might be seen in the examination of a reclining human body). The points are added to the 2D ellipse to define the location of the electrode (between the points) on the boundary of the ellipse.

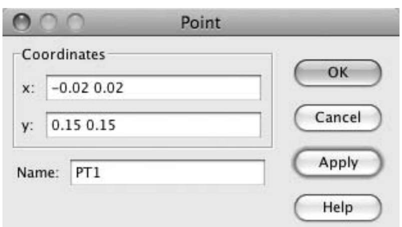


FIGURE 7.32 2D_EIS_2 model Point edit window

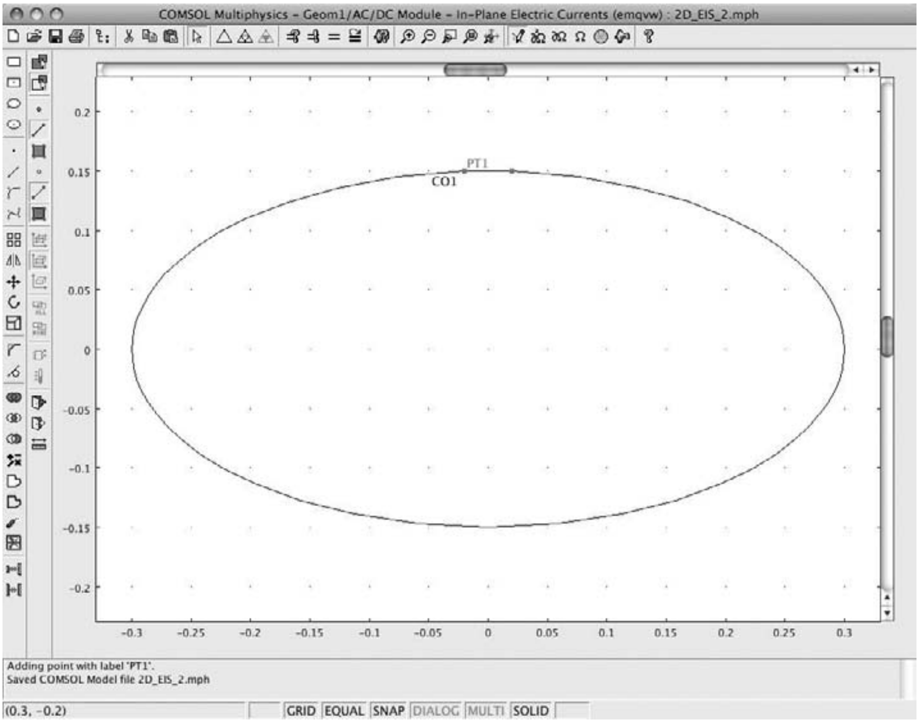


FIGURE 7.33 2D_EIS_2 model ellipse and points

Physics Settings: Scalar Expressions

Using the menu bar, select Options > Expressions > Scalar Expressions. In the Scalar Expressions edit window, enter the information shown in Table 7.6; also see Figure 7.34. Click OK.

Table 7.6 Scalar Expressions Edit Window

Name	Expression	Description
sigma_0	sig_bulk*(((x−x_0)^2+(y−y_0)^2)>r_00^2)	Conductivity bulk
epsilon_r_0	1+(eps_r_bulk-1)*(((x−x_0)^2+(y−y_0)^2)>r_00^2)	Permittivity relative
sigma_1	sig_bulk*(((x−x_1)^2+(y−y_1)^2)>r_01^2)	Conductivity bulk
epsilon_r_1	1+(eps_r_bulk-1)*(((x−x_1)^2+(y−y_1)^2)>r_01^2)	Permittivity relative
sigma_tot	(sigma_0+sigma_1)/2	Conductivity total
epsilon_r_tot	(epsilon_r_0+epsilon_r_1)/2	Permittivity total
r_00	r_0*(1.6-cos((t_0*pi)/8))/2	Radius ratio
r_01	r_1*(1.6-cos((t_0*pi)/8))/2	Radius ratio

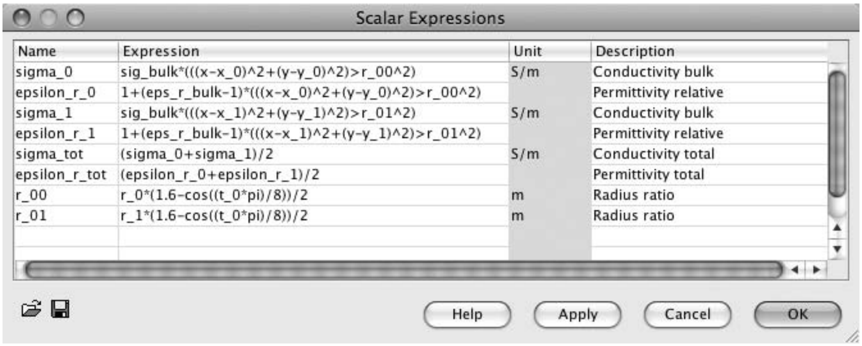


FIGURE 7.34 2D_EIS_2 model Scalar Expressions edit window

Physics Settings: Scalar Variables

Select Physics > Scalar Variables. Enter 1e6 in the nu_emqvw Application Scalar Variables edit window. See Figure 7.35. Click OK.

Physics Subdomain Settings: In-Plane Electric Currents (emqvw)

Having established the geometry for the 2D_EIS_2 model of an elliptical block with an electrode, the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). In the Subdomain edit windows, enter the information shown in Table 7.7.

Click the $D = \varepsilon_0\varepsilon_1 E$ radio button. See Figure 7.36. Click OK.

Physics Boundary Settings: In-Plane Electric Currents (emqvw)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 7.8. See Figures 7.37, 7.38, and 7.39.

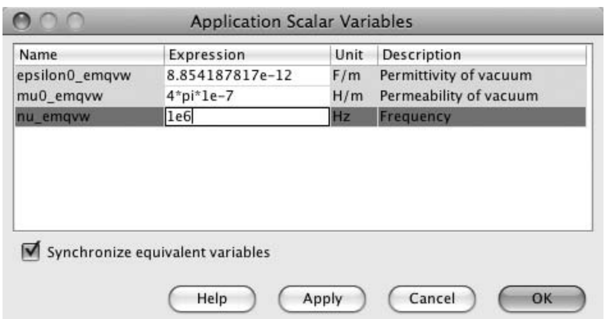


FIGURE 7.35 2D_EIS_2 model Application Scalar Variables edit window

Table 7.7 Subdomain Edit Window

Name	Expression	Description
σ (isotropic)	sigma_tot	Electrical conductivity
ε_r (isotropic)	epsilon_r_tot	Relative permittivity

Table 7.8 Boundary Settings – In-Plane Electric Currents (emqvw) Edit Window

Boundary	Boundary Condition	Figure Number
1, 4, 5, 8	Electric Insulation	7.37
2, 3	Port	7.38
6, 7	Ground	7.39

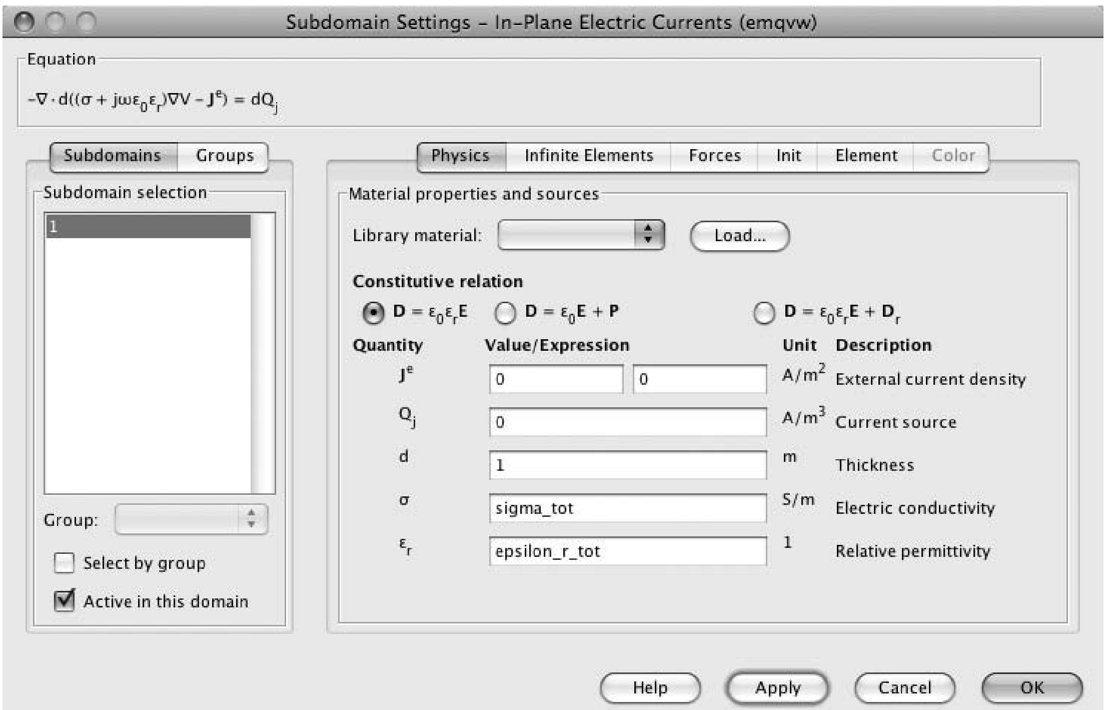


FIGURE 7.36 2D_EIS_2 model Subdomain Settings edit window

FIGURE 7.37 2D_EIS_2 model Boundary Settings (1, 4, 5, 8) edit window

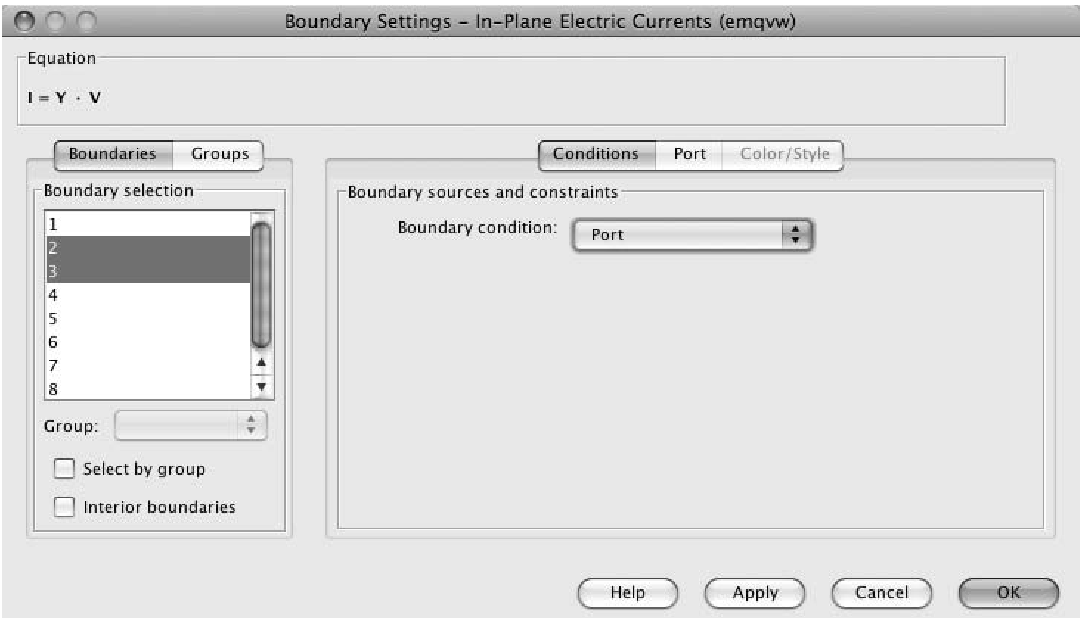


FIGURE 7.38 2D_EIS_2 model Boundary Settings (2, 3) edit window

FIGURE 7.39 2D_EIS_2 model Boundary Settings (6, 7) edit window

Select boundaries 2 and 3 in the Boundary selection window. Select the Port tab.

Check “Use port as input.” Select “Fixed current density” from the input property pull-down list; see Figure 7.40. Click OK.

FIGURE 7.40 2D_EIS_2 model Boundary Settings (Port) edit window

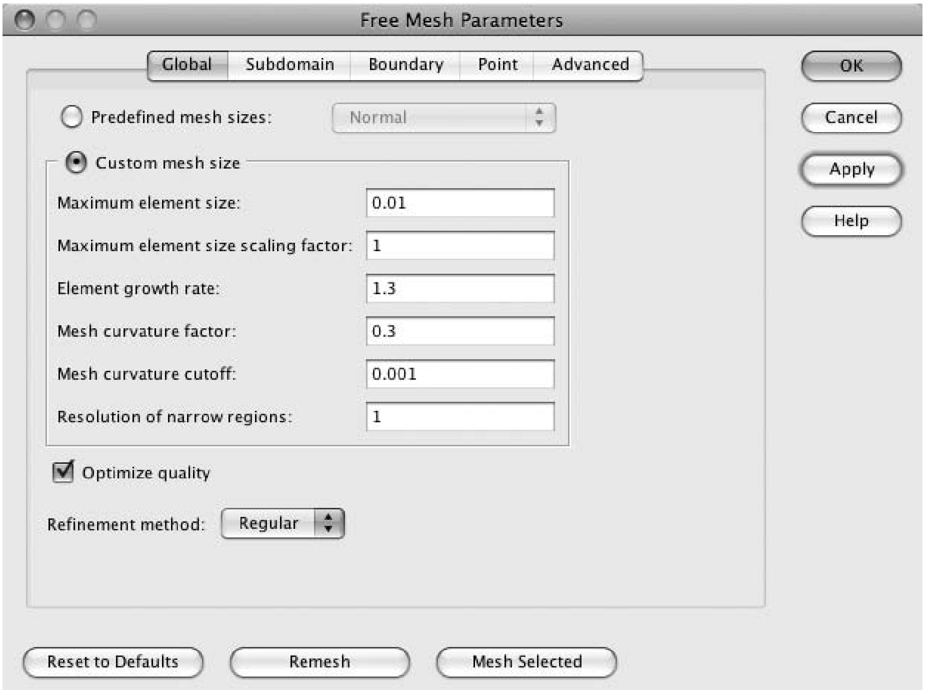


FIGURE 7.41 2D_EIS_2 model Free Mesh Parameters edit window

Mesh Generation

From the toolbar, select Mesh > Free Mesh Parameters > Global. Select Predefined mesh sizes > Normal (from the pull-down list). Select "Custom mesh size." Enter 0.01 in the Maximum element size edit window. See Figure 7.41.

Click the Remesh button, and then click OK. See Figure 7.42.

Solving the 2D_EIS_2 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Stationary solver.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. In this case, the modeler will need to change the chosen solver and the parametric settings.

Select "Parametric." Enter t_0 in the Parameter name edit window. Enter linspace(0.0,32,32) in the Parameter values edit window. See Figure 7.43. Click OK.

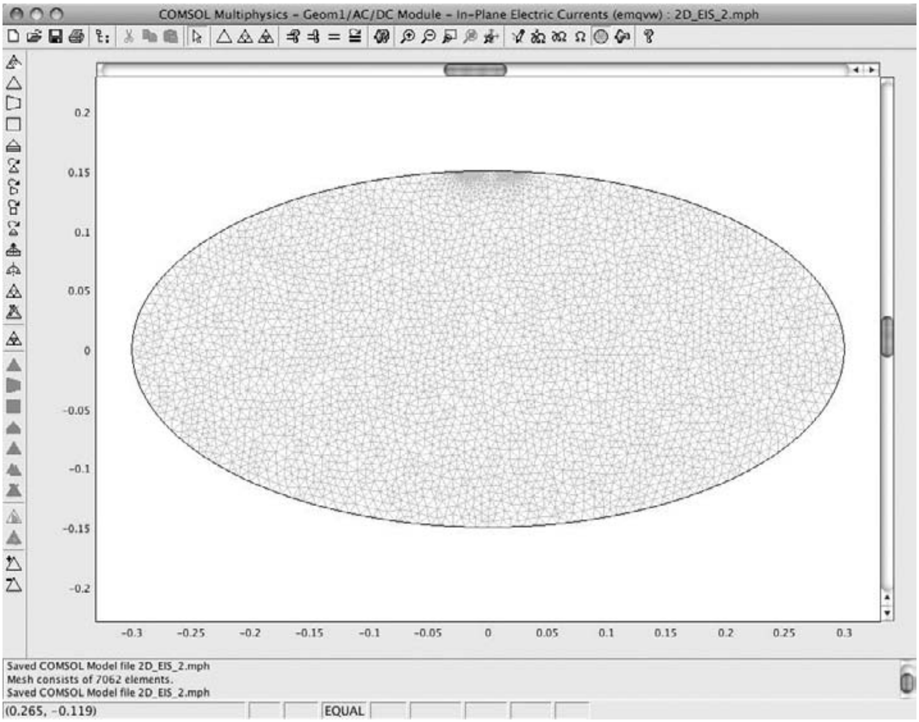


FIGURE 7.42 2D_EIS_2 model mesh

NOTE The linspace(0.0,32,32) command causes the solver to step the value of t_0 32 times between 0.0 and 32. For later versions of COMSOL Multiphysics software use the command range(0,32/32,32) in place of the linspace command.

In COMSOL Multiphysics software version 3.5a, the linspace (x_1,x_2,x_3) function (where x_1 = start value, x_2 = end value, and x_3 = number of intervals) has been changed to the range (y_1,y_2,y_3) function (where y_1 = start value, y_2 = interval width, and y_3 = end value).

Using the menu bar, Select Solve > Solve Problem. See Figure 7.44.

Postprocessing and Visualization

The default plot shows a surface plot of the electric potential (V) distribution in volts. To visualize the detected regions of differential conductivity, the plot parameters will need to be modified.

Select Postprocessing > Plot Parameters > Surface. Select "Total current density, norm" from the Predefined quantities pull-down list. Change the expression in the edit window from normJ_emqvw to 20*log10(normJ_emqvw). Click the Range button.

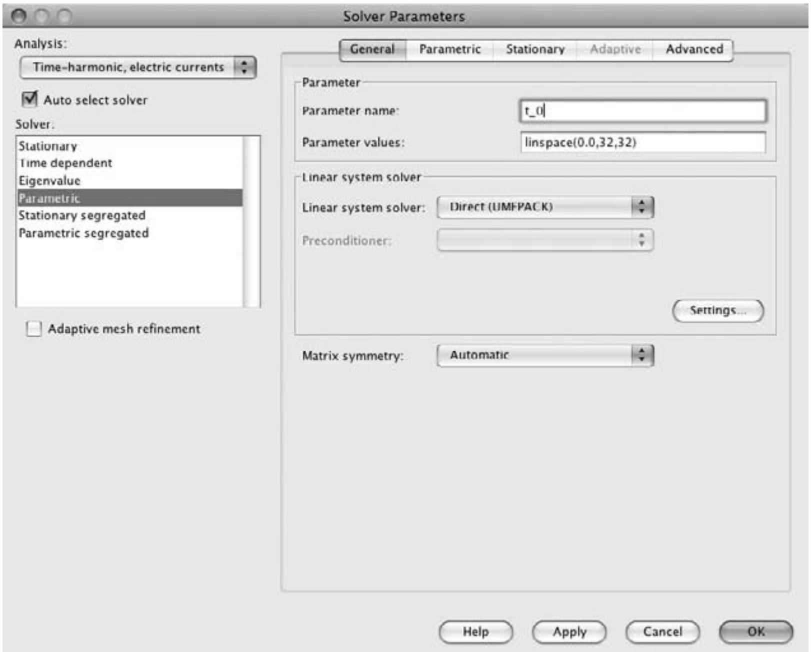


FIGURE 7.43 2D_EIS_2 model Solver Parameters edit window

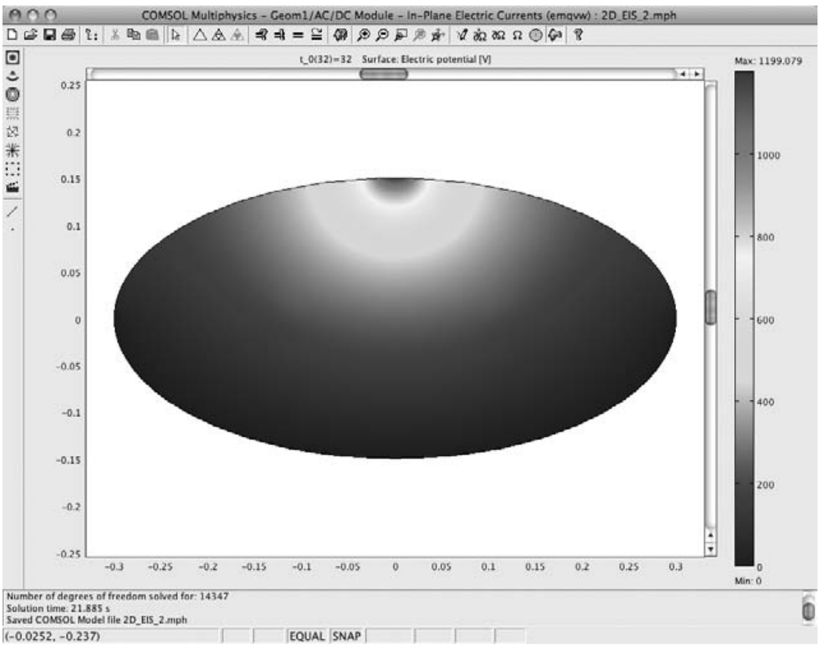


FIGURE 7.44 2D_EIS_2 model solution

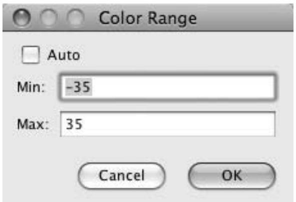


FIGURE 7.45 2D_EIS_2 model solution Color Range edit window

Unselect the Auto check box. Enter −35 in the Min edit window and 35 in the Max edit window. See Figure 7.45. Click OK.

Select "wave" from the Colormap pull-down list. See Figure 7.46.

Click OK. See Figure 7.47.

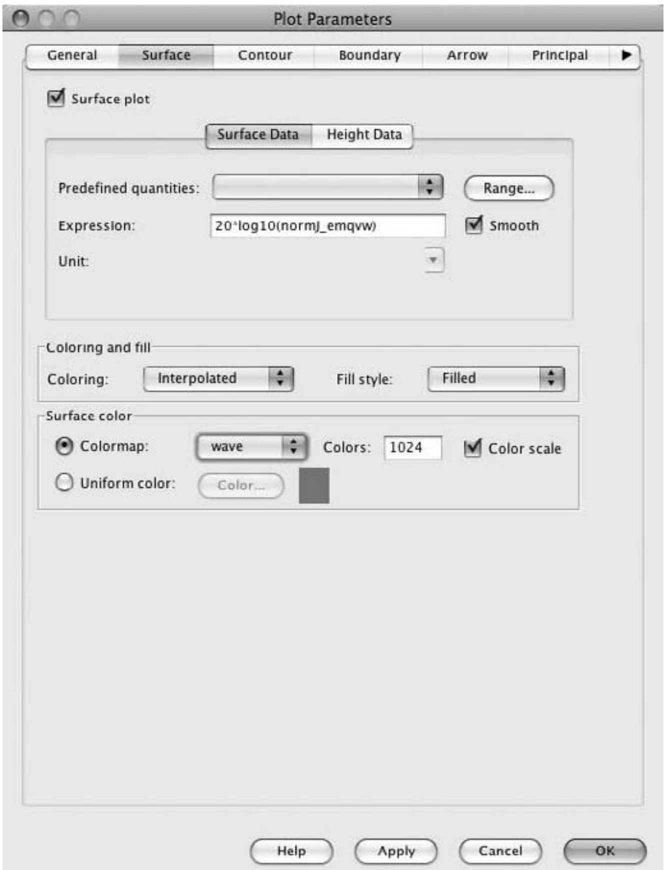


FIGURE 7.46 2D_EIS_2 model solution Plot Parameters edit window

FIGURE 7.47 2D_EIS_2 model solution with detected areas

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 7.48.

Click the Start Animation button. See Figure 7.49.

Alternatively, you can play the file Movie7_EIS_2.avi that was supplied with this book.

2D Electric Impedance Sensor Models: Summary and Conclusions

In this part of the chapter two 2D electric impedance sensor models, basic and advanced, were built and operated. These models employ a high-frequency current, (1 MHz alternating current AC) to explore the differential impedance within a body of material in a noninvasive manner. Such currents may be applied to the material of the modeled body to locate volumes that differ in impedance from the impedance of the bulk material by monitoring the local impedance. The basic 2D electric impedance sensor model shows the location of a fixed-volume impedance difference. The advanced 2D electric impedance sensor model shows the location of a fluctuating difference volume,

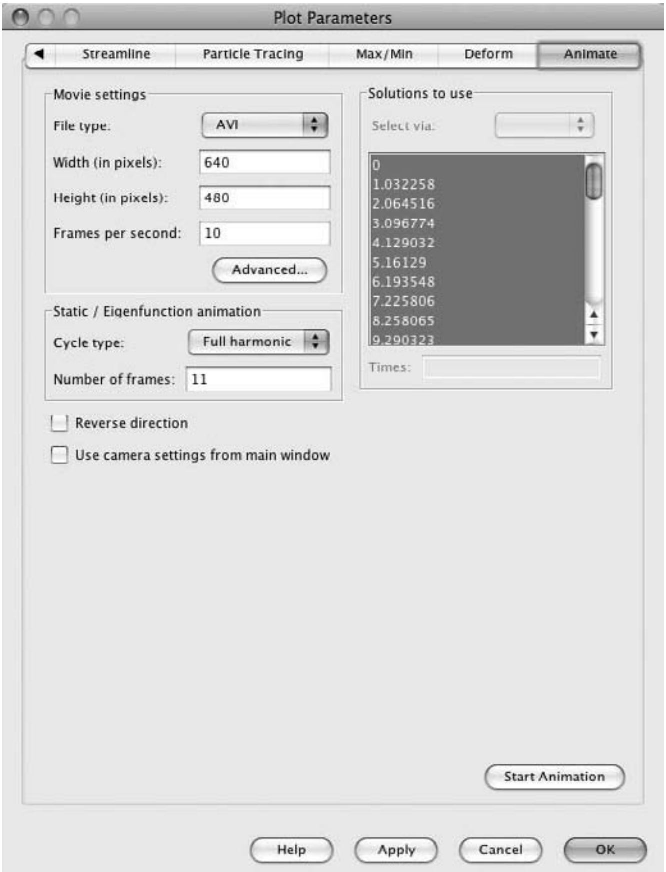


FIGURE 7.48 2D_EIS_2 model Plot Parameters window

as might be seen in a medical application measuring lung function. 2D electric impedance tomography research is currently exploring the application of this impedance-sensing measurement technology to the detection of breast cancer, lung function, brain function, and numerous other areas.

The new concepts introduced in this section of Chapter 7 are complex AC theory, complex impedance, and skin depth.

Generator and Power Distribution Basics

Shortly after Georg Ohm discovered and published Ohm's law in 1827, Michael Faraday[19] discovered and published the basic operating principle of both DC and AC generators, known as electromagnetic induction.[20]

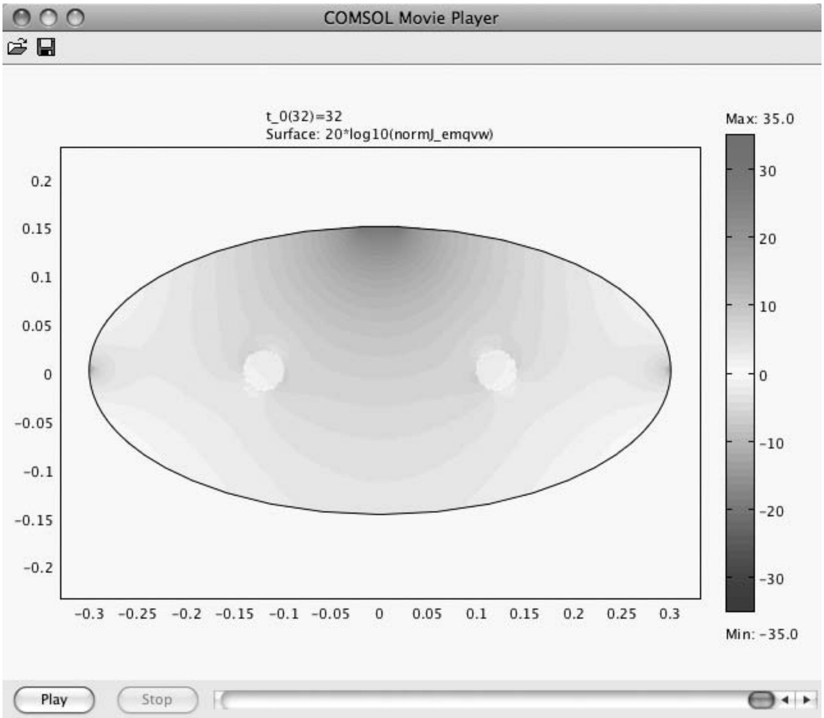


FIGURE 7.49 2D_EIS_2 model animation, final frame

NOTE Ohm's law is:

$$I = \frac{E}{R} \tag{7.10}$$

where I = current in amperes (A)
E = electromotive force in volts (V)
R = resistance in ohms

Thomas Alva Edison[21] took the initial lead in the development and commercialization of DC electrical power generation and distribution systems. During approximately the same time period, George Westinghouse[22] and Nikola Tesla[23] were developing and commercializing AC electrical power generation and distribution systems. The resulting intense industrial competition led to what has been called the "War of Currents."[24]

Independent of all the rhetoric exchanged during the "War of Currents," two fundamental physical factors would mandate that the ultimate winner of this intense contest was to be AC power, even before the first battle was fought. Those basic physical factors were (1) the intrinsic nature of DC (steady) and (2) Joule's first law.[25] It is the

intrinsic nature of DC that it is, by definition, a steady, fixed voltage. Thus it is by definition not transformable to a different voltage. For DC to be transformable, it must be converted to AC, transformed, and then converted back to DC. Therefore, when different voltages were needed, different dynamos (DC generators) had to be built to generate the different voltage.

Joule's first law, published in 1841, states that the power dissipated in a resistor can be expressed as follows:

$$P = I^2{*}R \tag{7.11}$$

By Ohm's law

$$R = \frac{E}{I} \tag{7.12}$$

Thus

$$P = I^2{*}\left(\frac{E}{I}\right) = I{*}E$$

It is the intrinsic nature of AC that both the current and the voltage normally fluctuate. Hence, AC can be converted (transformed) from one voltage to a different voltage.[26] Because AC can be transformed to the first order (assuming no systemic losses) and assuming conservation of energy (no sources or sinks in the transformation process), then

---

NOTE The term "to the first order" means formulating the most basic mathematical statement of the problem, without secondary corrections.

Primary input power: $P_P = I_P{*}E_P$ (7.13)

Secondary output power: $P_S = I_S{*}E_S$ (7.14)

Conservation of energy: $P_P = P_S$ (7.15)

where P_P = power input to the transformer in watts (W)
E_P = electromotive force input to the transformer in volts (V)
I_P = current input to the transformer in amperes (A)
P_S = power output from the transformer in watts (W)
E_S = electromotive force output from the transformer in volts (V)
I_S = current output from the transformer in amperes (A)

Assuming a lossless transformer,

$$E_S = \frac{N_S}{N_P} * E_P \tag{7.16}$$

where E_P = electromotive force at the primary input of the transformer in volts (V)
E_S = electromotive force at the secondary output of the transformer in volts (V)
N_P = number of turns in the primary winding of the transformer
N_S = number of turns in the secondary winding of the transformer

Because the input power equals the output power, the current (I) and the electromotive force (E) have an inverse relationship. As E goes up, I goes down, in a direct proportionality.

Thus

$$I_S = \frac{N_P}{N_S} * I_P \tag{7.17}$$

where I_P = current in the primary winding of the transformer in amperes (A)
I_S = current in the secondary winding of the transformer in amperes (A)
N_P = number of turns in the primary winding of the transformer
N_S = number of turns in the secondary winding of the transformer

In the case of a DC distribution system, as shown in Figure 7.50, the transmission line has losses.

In the case of DC power, all of the load current flows through the transmission line resistance and generates thermal losses:

$$P_{DC\,Loss} = I^2_{DC\,Load} * R_{TL} \tag{7.18}$$

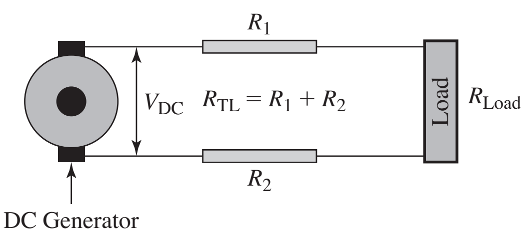


FIGURE 7.50 DC power transmission system

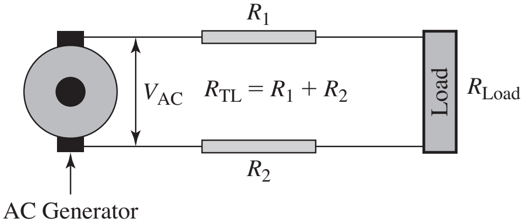


FIGURE 7.51 Untransformed AC power transmission system

If the AC power transmission system were configured in the same manner as a DC power transmission system, then the systems would be equivalent. See Figure 7.51.

However, when transformers are employed, the physics changes significantly. See Figure 7.52.

In the case of transformed AC power, for example, raising the AC electromotive force (EMF) of the transmission line by transforming the EMF by a factor of 100 causes the current in the transmission line to be lowered by a factor of 100:

$$\frac{E_S}{E_P} = \frac{N_S}{N_P} = 100 \tag{7.19}$$

Then

$$I_{ACTL} = \frac{1}{100} * I_{ACLoad} \tag{7.20}$$

and

$$P_{ACLoss} = \left(\frac{I_{ACLoad}}{100}\right)^2 * R_{TL} \tag{7.21}$$

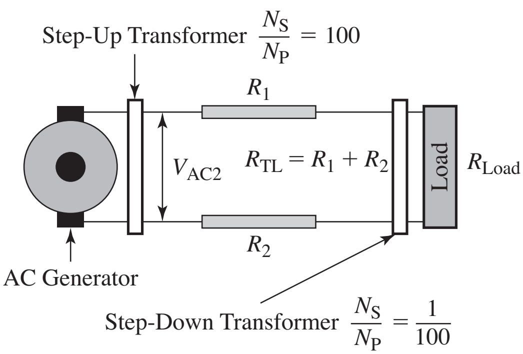


FIGURE 7.52 Transformed AC power transmission system

where P_{DS} = dissipated power in watts (W)
E_P = electromotive force at the primary input of the transformer in volts (V)
E_S = electromotive force at the secondary output of the transformer in volts (V)
N_P = number of turns in the primary winding of the transformer
N_S = number of turns in the secondary winding of the transformer
$I_{AC\ TL}$ = transmission line current
$I_{AC\ Load}$ = AC load current
$P_{AC\ Loss}$ = transmission line current
R_{TL} = transmission line resistance

Assuming that the load currents are equivalent

$$I_{DCLoad} = I_{ACLoad} \tag{7.22}$$

then the relative transmission line power loss for AC compared to DC is

$$\frac{P_{ACLoss}}{P_{DCLoss}} = \left(\frac{1}{100}\right)^2 = 1*10^{-4} \tag{7.23}$$

Thus transformed AC became the obvious choice for power line transmission, based both on versatility and on reduced power losses.

2D AC Generators: Static and Transient

In the following subsections of Chapter 7, models are developed that provide an analysis of the rotating machines (AC generators) that convert mechanical energy into electrical energy (AC power). The generation of AC power is accomplished through the application of Faraday's law of induction. In the following models, a magnetic vector potential (**A**) is employed that has only a *z* component.

Rotation is modeled using the deformed Mesh Application Mode (ALE). The rotor and the stator are drawn separately and then combined as an assembly.[27]

The materials employed in this model are high-energy samarium–cobalt magnets with nonlinear soft iron pole pieces.

NOTE A pole piece is the magnetically soft (easily altered) material that is inserted in the magnet circuit to guide the path of the magnetic flux to a desired location.

2D AC Generator Model (2D_ACG_1): Static

The numerical solution model developed in this section (2D_ACG_1) is derived from a model that was originally developed by COMSOL as an AC/DC Module Motors and Drives Library Model. Here, the 2D generator model (2D_ACG_1) will be built as a

FIGURE 7.53 2D_ACG_1 Model Navigator setup

static (stationary) model. In the next subsection, the static model will be used as the starting point for the transient (rotating) model (2D_ACG_2).

To start building the 2D_ACG_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from the Space dimension pull-down list. Select AC/DC Module > Rotating Machinery > Rotating Perpendicular Currents. See Figure 7.53. Click OK.

NOTE The Model Navigator shows two names in the Application Mode name edit window: emqa and ale.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 7.9; see also Figure 7.54. Click OK.

NOTE When the modeler enters the constant t, the text will become red in color to indicate that the modeler has entered a reserved variable name (t = time, in COMSOL Multiphysics software). However, because the first model is stationary, t needs to be assigned a value (in this case, 0). Once the transient model is built, the transient solver will override the assigned value during the solving process.

Table 7.9 Constants Edit Window

Name	Expression	Description
t	0[s]	Time equals zero (stationary solution)
rpm	60[1/min]	Revolutions per minute
A	pi*(0.02[m])^2	Area, stator wire
L	0.4[m]	Length, generator
NN	1	Stator winding turns

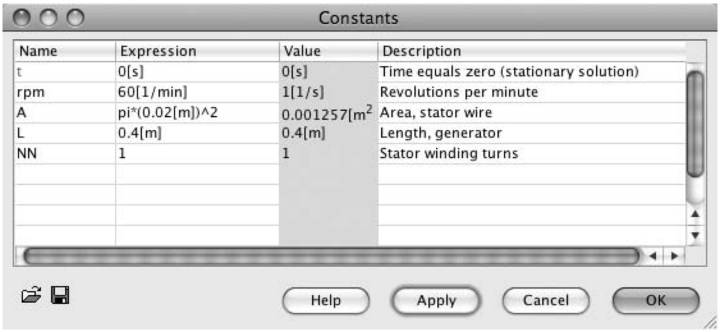


FIGURE 7.54 2D_ACG_1 model Constants edit window

Select File > Save As. Enter 2D_ACG_1 in the Save As edit window. See Figure 7.55. Click the Save button.

Generator Geometry

NOTE The 2D_ACG_1 geometry is very complex. Be sure to follow the steps carefully and in sequence. After completion of all of the geometry and assembly steps, there should be a total of 152 boundaries.

Using the menu bar, select Draw > Specify Objects > Circle and create the circles indicated in Table 7.10.

Select File > Save. See Figure 7.56.

Table 7.10 Stator Geometry Circles Creation

Name	Radius	Base	*X*	*Y*	Rotation Angle
C1	0.3	Center	0	0	22.5
C2	0.235	Center	0	0	0
C3	0.225	Center	0	0	0
C4	0.4	Center	0	0	0

FIGURE 7.55 2D_ACG_1 model Save As edit window

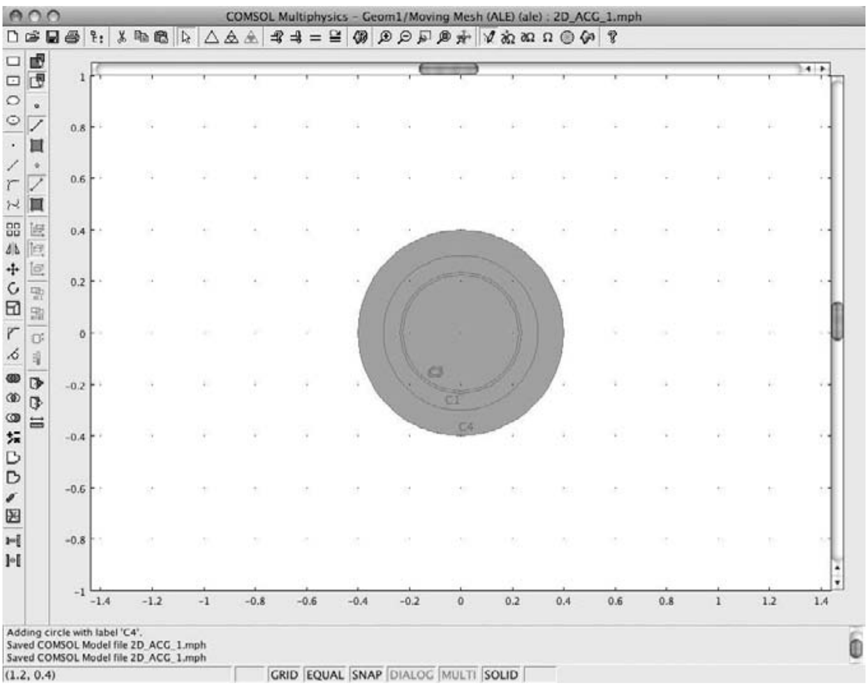


FIGURE 7.56 2D_ACG_1 model created circles

Table 7.11 Geometry Rectangles Creation

Name	Width	Height	Base	*X*	*Y*	Rotation Angle
R1	0.1	1.0	Center	0	0	0
R2	0.1	1.0	Center	0	0	45
R3	0.1	1.0	Center	0	0	90
R4	0.1	1.0	Center	0	0	135

Using the menu bar, select Draw > Specify Objects > Rectangle and create the rectangles indicated in Table 7.11.

Select File > Save. See Figure 7.57.

Select Draw > Create Composite Object. Uncheck the Keep interior boundaries check box. Enter C2+C1*(R1+R2+R3+R4) in the Set formula edit window. See Figure 7.58.

NOTE The commands +, *, and – equal union, intersection, and difference, respectively. Enter the formulas *exactly* as indicated, or the resulting geometry will be incorrect.

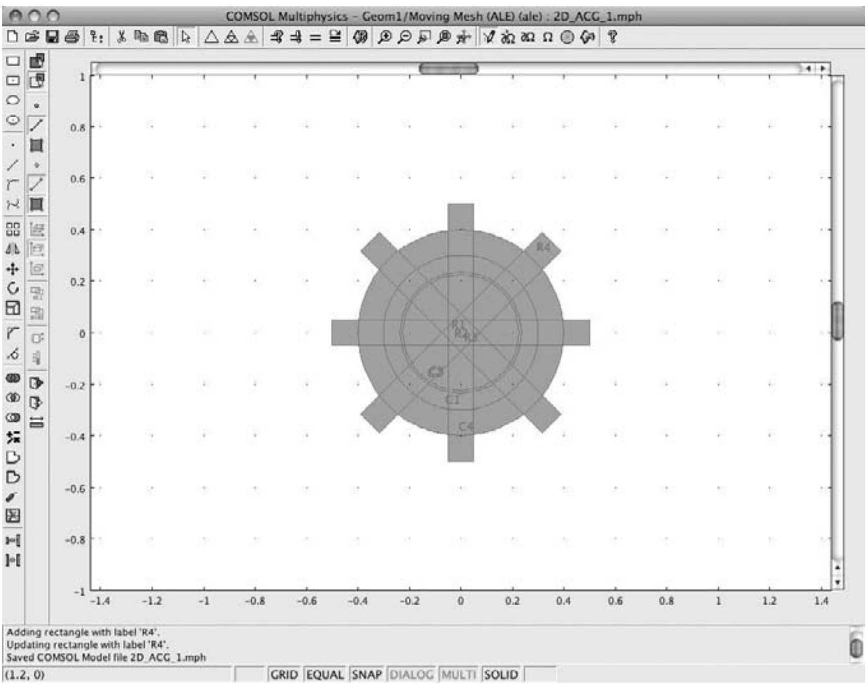


FIGURE 7.57 2D_ACG_1 model created circles and rectangles

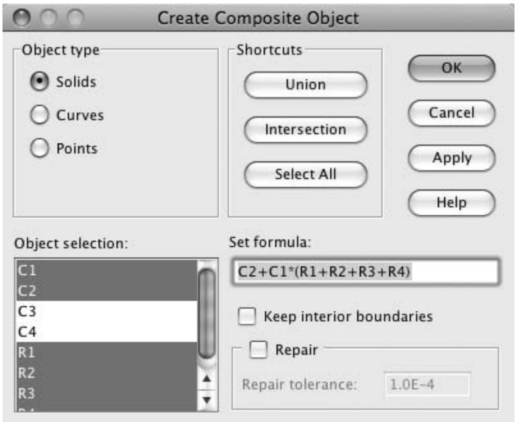


FIGURE 7.58 2D_ACG_1 model Create Composite Object edit window

Click OK. See Figure 7.59.

Select Draw > Create Composite Object. Check the Keep interior boundaries check box. Enter C4+CO1–C3 in the Set formula edit window. See Figure 7.60.

Click OK. See Figure 7.61.

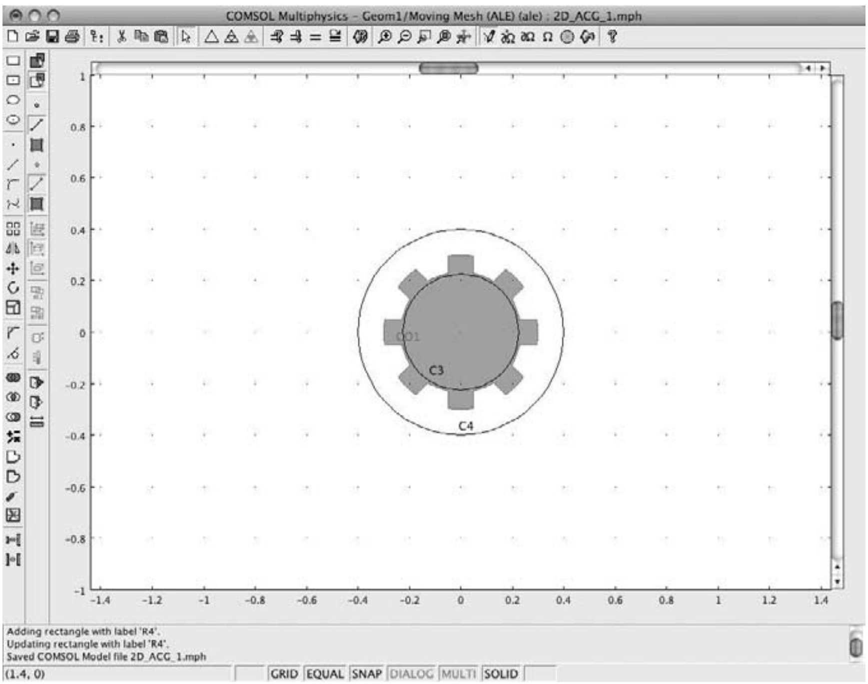


FIGURE 7.59 2D_ACG_1 model CO1

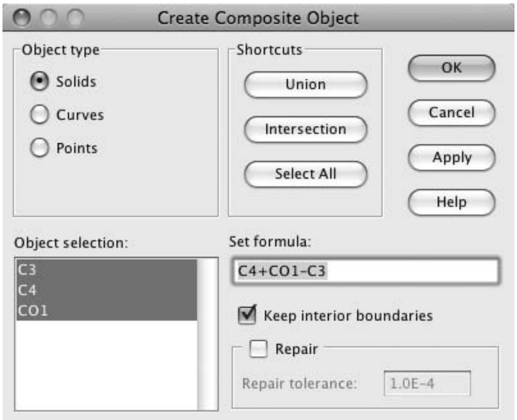


FIGURE 7.60 2D_ACG_1 model Create Composite Object edit window

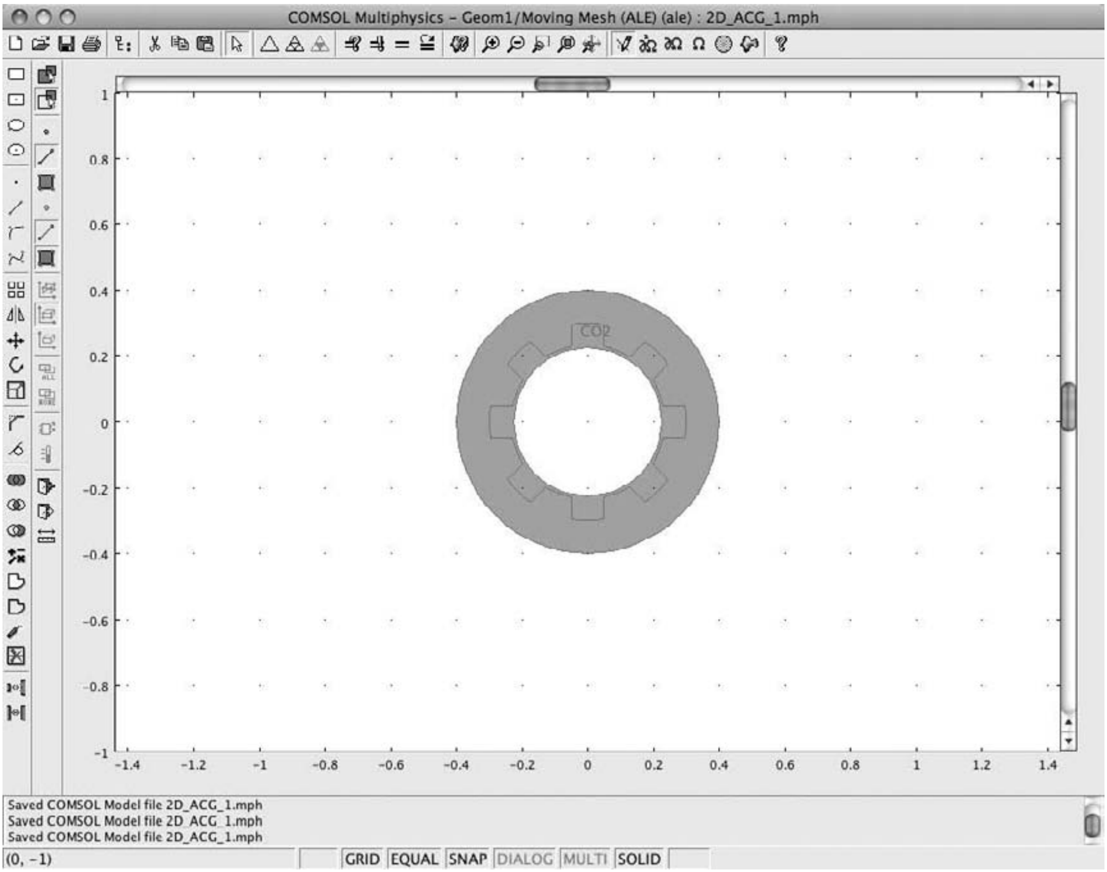


FIGURE 7.61 2D_ACG_1 model CO2

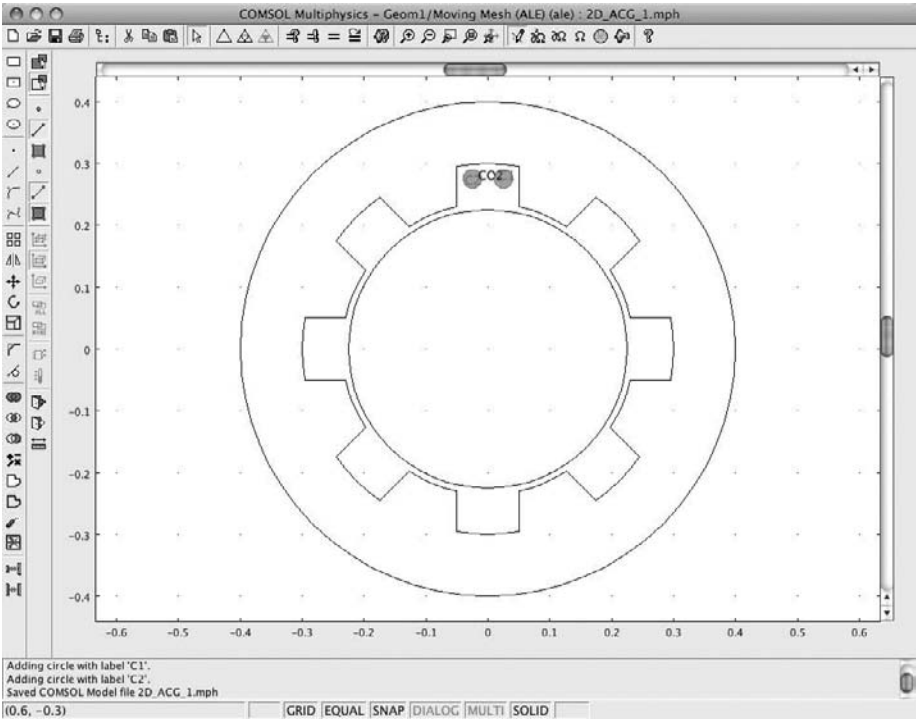


FIGURE 7.62 2D_ACG_1 model two new created circles

Click the Zoom Extents button. Using the menu bar, select Draw > Specify Objects > Circle and create the two new circles indicated in Table 7.12.

Click the Save button. See Figure 7.62.

Select C1 and C2. Using the menu bar, select Edit > Copy. Using the menu bar, select Edit > Paste. Verify that the displacements are X:0 and Y:0. See Figure 7.63. Click OK.

Using the menu bar, select Draw > Modify > Rotate. Enter 45 in the Rotation angle edit window. See Figure 7.64. Click OK.

Select C1 and C2. Using the menu bar, select Edit > Copy. Using the menu bar, select Edit > Paste. Verify that the displacements are X:0 and Y:0. Click OK.

Table 7.12 Geometry Circles Creation

Name	Radius	Base	*X*	*Y*	Rotation Angle
C1	0.015	Center	0.025	0.275	0
C2	0.015	Center	−0.025	0.275	0

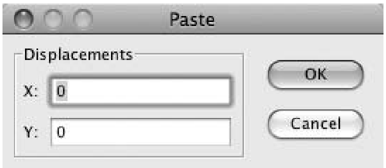


FIGURE 7.63 2D_ACG_1 model Paste C1 and C2

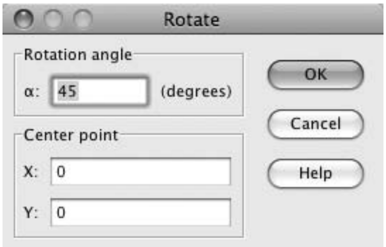


FIGURE 7.64 2D_ACG_1 model rotated Paste C3 and C4

Using the menu bar, select Draw > Modify > Rotate, for the indicated angles for each circle pair shown in Table 7.13. See Figure 7.65.

Select Edit > Select All. Select Draw > Create Composite Object. Click the Union button and then click OK. See Figure 7.66.

NOTE The stationary portion (stator) of the generator has now been created. The rotating portion (rotor) will be created next.

Using the menu bar, select Draw > Specify Objects > Circle and create the circles indicated in Table 7.14.

Click the Save button. See Figure 7.67.

Table 7.13 Geometry Circles: Copy, Rotate, and Paste

Name	Rotation Angle
C5, C6	90
C7, C8	135
C9, 10	180
C11, C12	−45
C9, 10	−90
C11, C12	−135

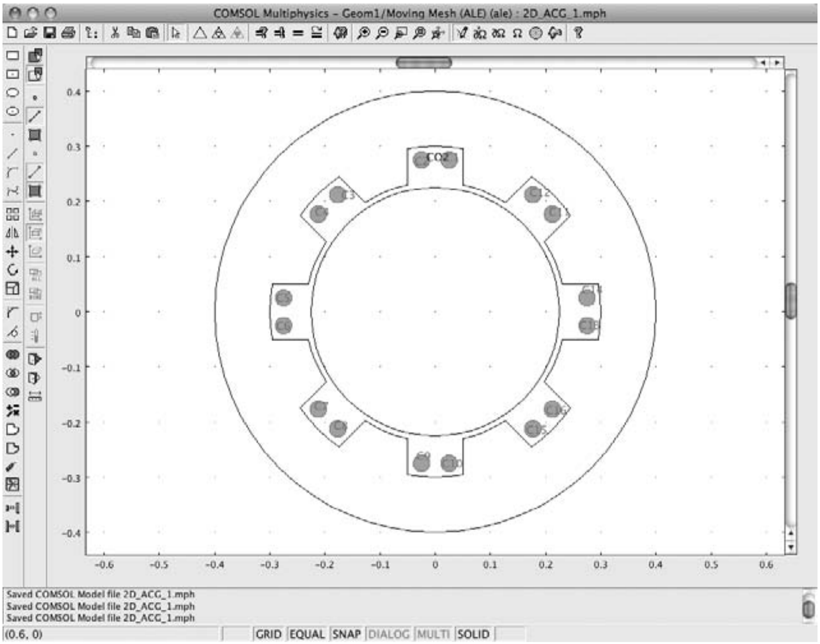


FIGURE 7.65 2D_ACG_1 model rotated, pasted circles

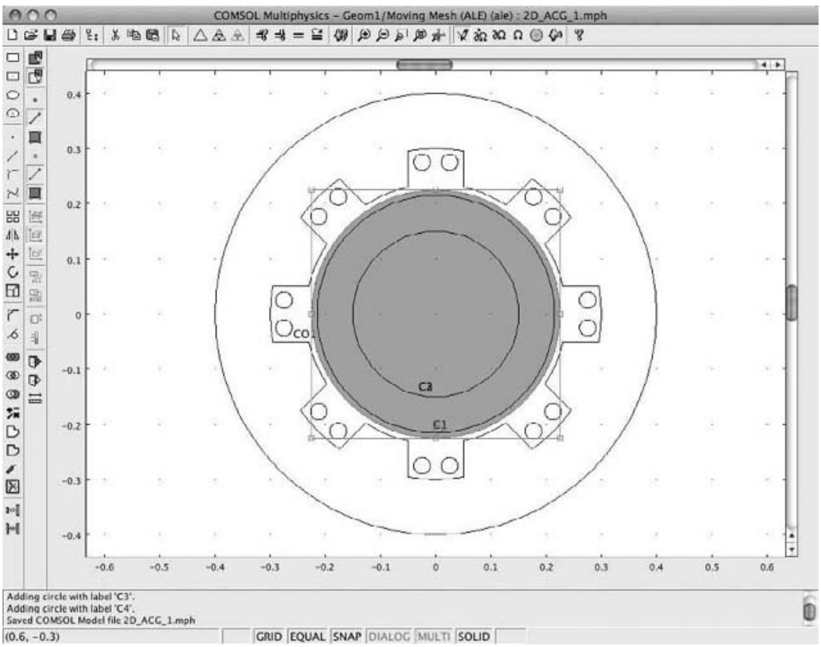


FIGURE 7.66 2D_ACG_1 model union of all objects, CO1

Table 7.14 Rotor Geometry Circles Creation

Name	Radius	Base	*X*	*Y*	Rotation Angle
C1	0.215	Center	0	0	0
C2	0.15	Center	0	0	22.5
C3	0.15	Center	0	0	22.5
C4	0.225	Center	0	0	0

Using the menu bar, select Draw > Specify Objects > Rectangle and create the rectangles indicated in Table 7.15.

Click the Save button. See Figure 7.68.

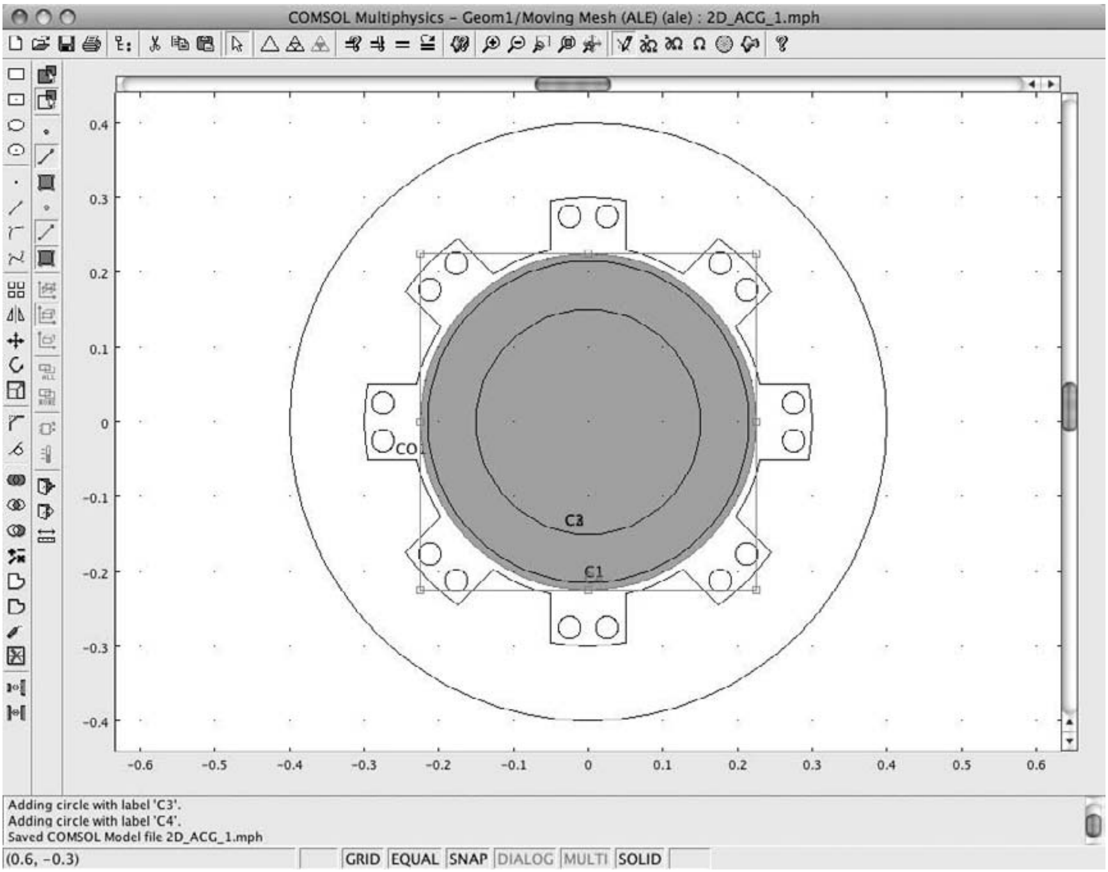


FIGURE 7.67 2D_ACG_1 model rotor-created circles

Table 7.15 Rotor Geometry Rectangles Creation

Name	Width	Height	Base	*X*	*Y*	Rotation Angle
R1	0.1	1.0	Center	0	0	22.5
R2	0.1	1.0	Center	0	0	−22.5
R3	0.1	1.0	Center	0	0	67.5
R4	0.1	1.0	Center	0	0	−67.5

Select Draw > Create Composite Object. Uncheck the Keep interior boundaries check box. Enter C2+C1*(R1+R2+R3+R4) in the Set formula edit window. See Figure 7.69. Click OK.

NOTE The commands +, *, and – equal union, intersection, and difference, respectively. Enter the formulas *exactly* as indicated, or the resulting geometry will be incorrect.

FIGURE 7.68 2D_ACG_1 model rotor-created rectangles

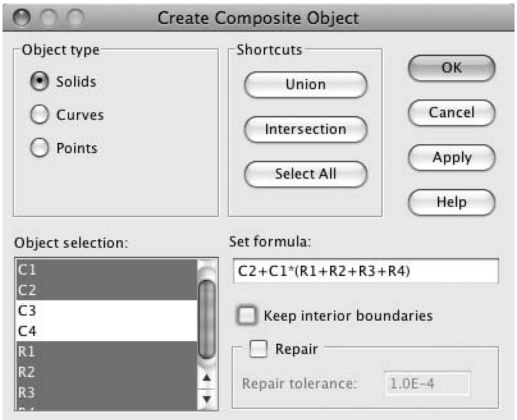


FIGURE 7.69 2D_ACG_1 model Create Composite Object edit window

Select Draw > Create Composite Object. Check the Keep interior boundaries check box. Enter CO2+C3+C4 in the Set formula edit window. See Figure 7.70.

Click OK. See Figure 7.71.

NOTE The names Stator and Rotor can be assigned to the appropriate created composite objects by clicking on the composite object (selecting) and selecting Draw > Object Properties. You can then enter the chosen name (Stator, Rotor) in the Name edit window.

Select File > Export > Geometry Objects to File. Enter 2D_ACG_1_Geometry in the Save As edit window. Select "COMSOL Multiphysics binary file (*.mphbin)" from the File Format pull-down list. Click the Save button. See Figure 7.72.

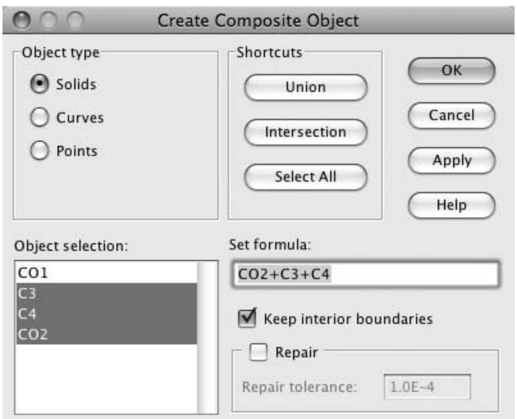


FIGURE 7.70 2D_ACG_1 model Create Composite Object edit window

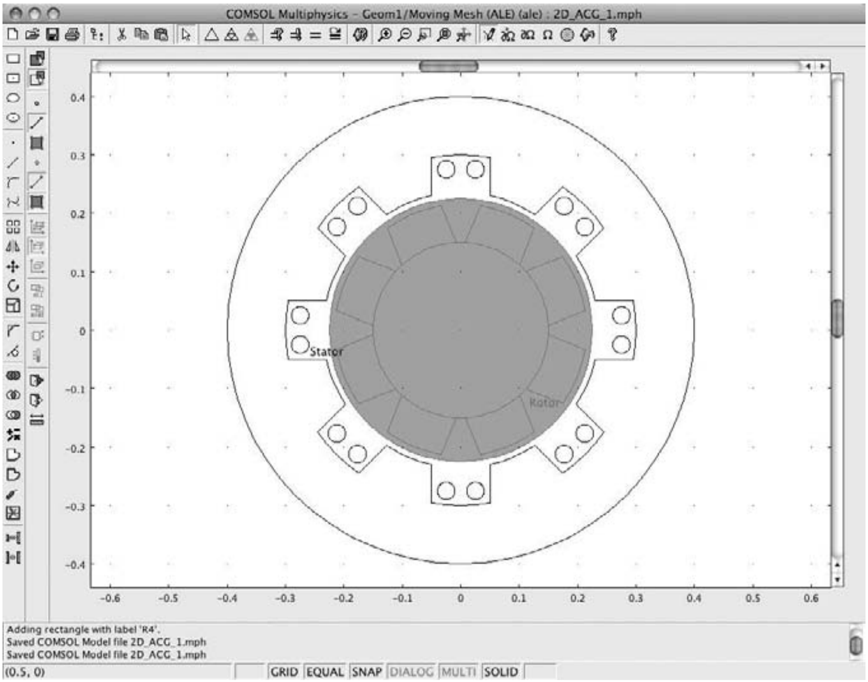


FIGURE 7.71 2D_ACG_1 model composite objects

FIGURE 7.72 2D_ACG_1 model stator and rotor composite objects

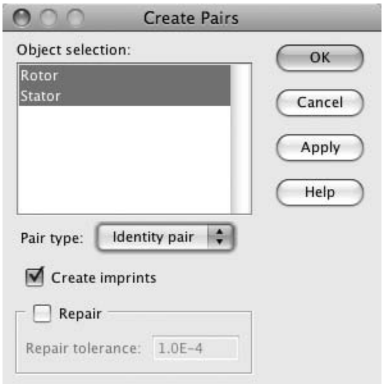


FIGURE 7.73 2D_ACG_1 model stator and rotor Create Pairs edit window

Assemble the Generator Geometry (Stator and Rotor)

Using the menu bar, select Draw > Create Pairs. Select both objects (Rotor and Stator). Select "Identity pair" from the pair type pull-down list. See Figure 7.73.

Click OK. See Figure 7.74.

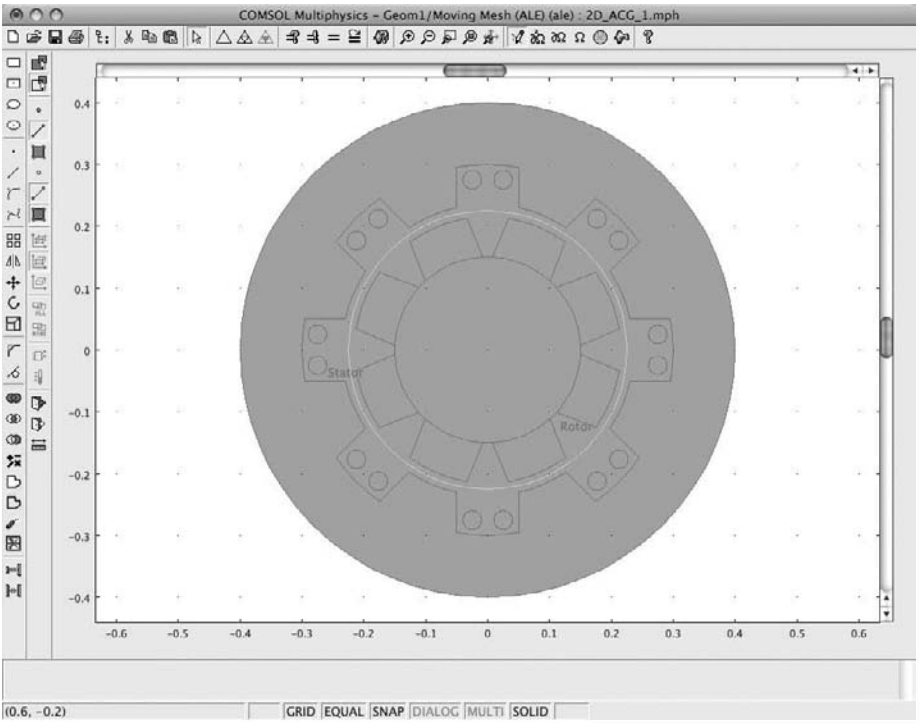


FIGURE 7.74 2D_ACG_1 model paired stator and rotor

Table 7.16 Subdomain Integration Variables Edit Window

Subdomains	Name	Expression	Integration Order	Global Destination
3, 4, 9–12, 17, 18	Vi	L*Ez_emqa/A	4	Yes
5–8, 13–16	Vi	−L*Ez_emqa/A	4	Yes

NOTE The pairing of the rotor and stator couples the boundaries of the two separately created geometric elements. This pairing facilitates the use of the sliding mesh at the boundary between the rotor and the stator, which would otherwise be discontinuous.

Physics Settings: Subdomain Integration Variables

Using the menu bar, select Options > Integration Coupling Variables > Subdomain Variables. In the Subdomain Integration Variables edit window, enter the information shown in Table 7.16; also see Figure 7.75. Click OK.

NOTE The integration variable expressions for Vi are summed to yield the voltage induced into the windings of the generator.

Select Options > Materials/Coefficients Library. Verify that the Electric (AC/DC) Materials Properties Library is loaded. If not, you will need to load that library to complete this model. See Figure 7.76. Click OK.

Physics Subdomain Settings: Perpendicular Induction Currents, Vector Potential (emqa)

Having established the geometry for the 2D_ACG_1 model, the next step is to define the fundamental Physics conditions. Using the menu bar, select Multiphysics >

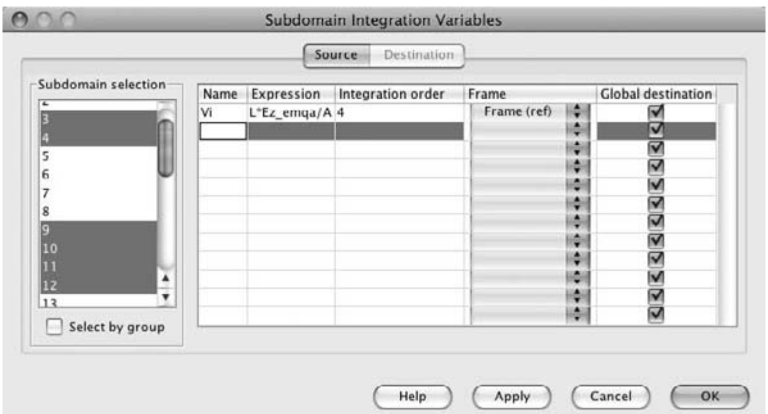


FIGURE 7.75 2D_ACG_1 model Subdomain Integration Variables edit window

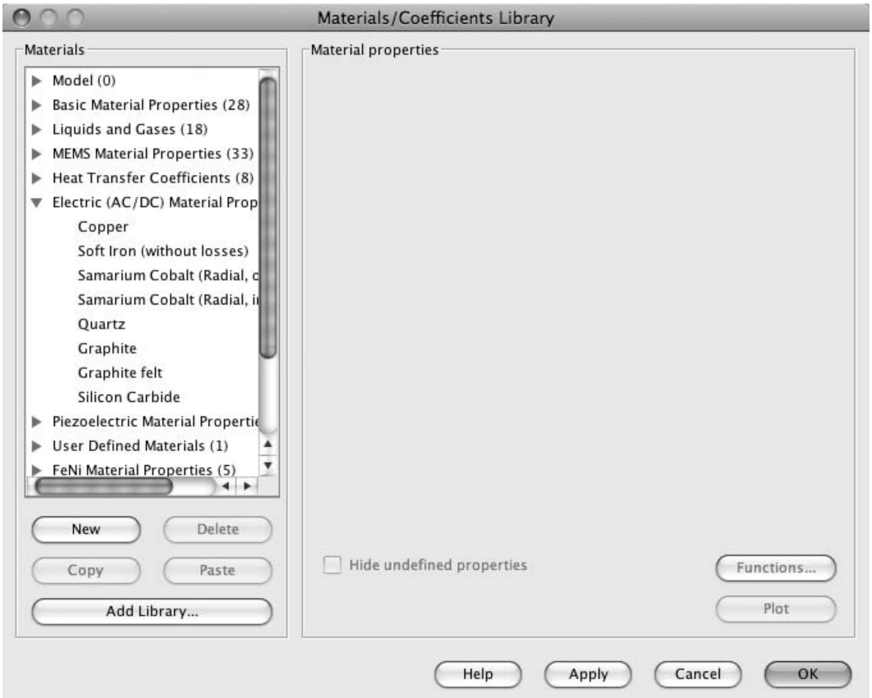


FIGURE 7.76 Materials/Coefficients Library window

Perpendicular Induction Currents, Vector Potential (emqa). Using the menu bar, select Physics > Properties. In the Application Mode Properties dialog box, choose "On" from the Weak constraints pull-down list. See Figure 7.77. Click OK.

Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window. Click the Load button.

FIGURE 7.77 2D_ACG_1 model Application Mode Properties dialog box

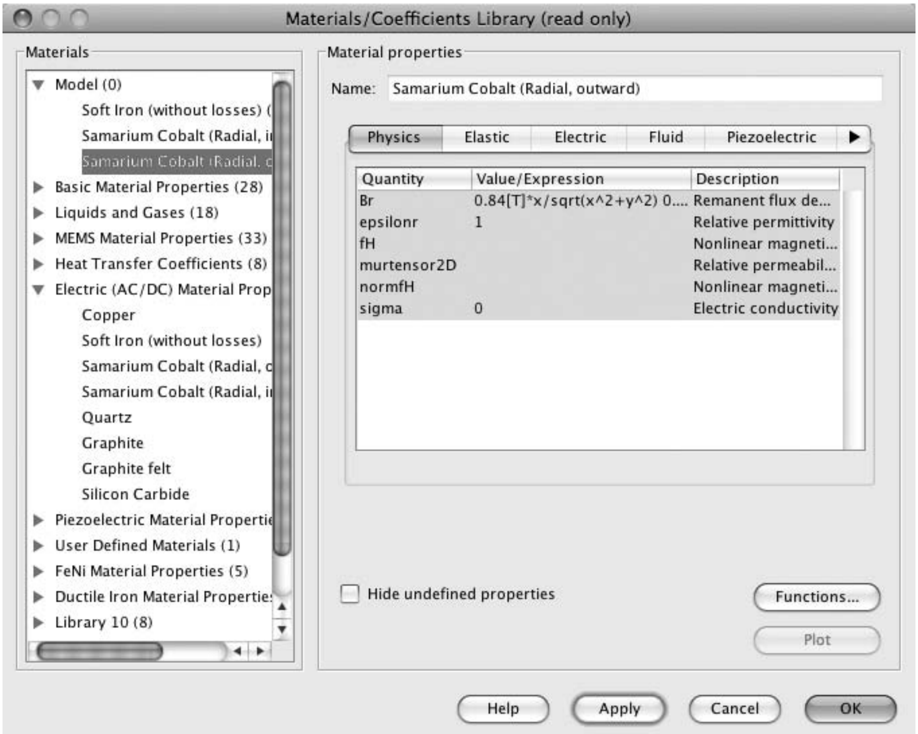


FIGURE 7.78 2D_ACG_1 model Materials/Coefficients Library load window

Select Electric (AC/DC) Materials Properties Library > Soft Iron (without losses). Click the Apply button.

Select Electric (AC/DC) Materials Properties Library > Samarium Cobalt (Radial, inward). Click the Apply button.

Select Electric (AC/DC) Materials Properties Library > Samarium Cobalt (Radial, outward). Click the Apply button. See Figure 7.78. Click the Cancel button.

NOTE The last three commands added the three selected materials to the Model (0) Library for use in this model.

In the Subdomain edit windows, enter the information shown in Table 7.17. Click OK. See Figures 7.79–7.82.

Physics Subdomain Settings: Moving Mesh (ALE) (ale)

Using the menu bar, select Physics > Multiphysics > Moving Mesh (ALE) (ale). Using the menu bar, select Physics > Subdomain Settings. In the Subdomain edit windows, enter the information shown in Table 7.18. Click OK. See Figure 7.83.

Table 7.17 Subdomain Edit Window

Subdomain	H ⇔ B Setting	Material	Figure Number
1, 3–19	$B = \mu_0\mu_r H$	—	7.79
2, 28	$H = f(\|B\|)e_B$	Soft iron (without losses)	7.80
20, 23, 24, 27	$B = \mu_0\mu_r H + B_r$	Samarium cobalt (radial, inward)	7.81
21, 22, 25, 26	$B = \mu_0\mu_r H + B_r$	Samarium cobalt (radial, outward)	7.82

Table 7.18 Subdomain Settings – Moving Mesh Edit Window

Subdomain	Group	Figure Number
19–28	rotate_CCW	7.83

Physics Boundary Settings

Leave the Boundary Settings at the default conditions. The identity pair couples the stator and the rotor.

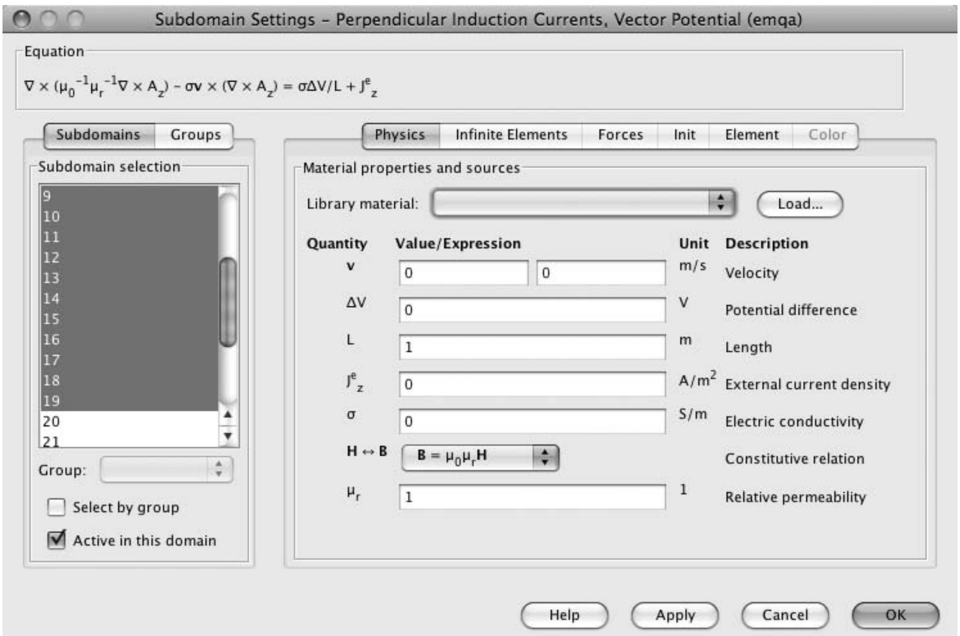


FIGURE 7.79 2D_ACG_1 model Subdomain Settings (1, 3–19) edit window

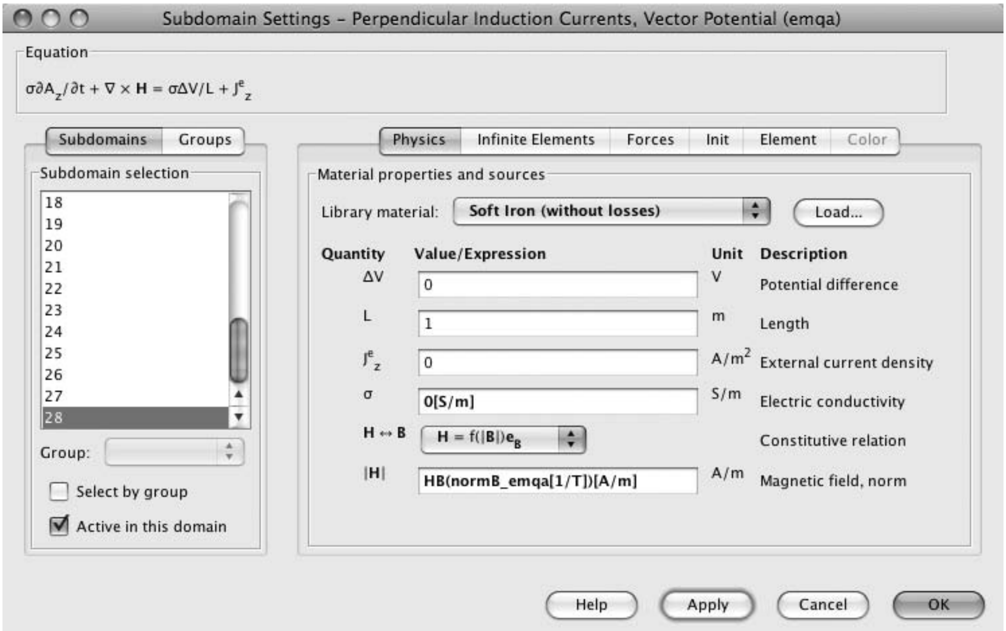


FIGURE 7.80 2D_ACG_1 model Subdomain Settings (2, 28) edit window

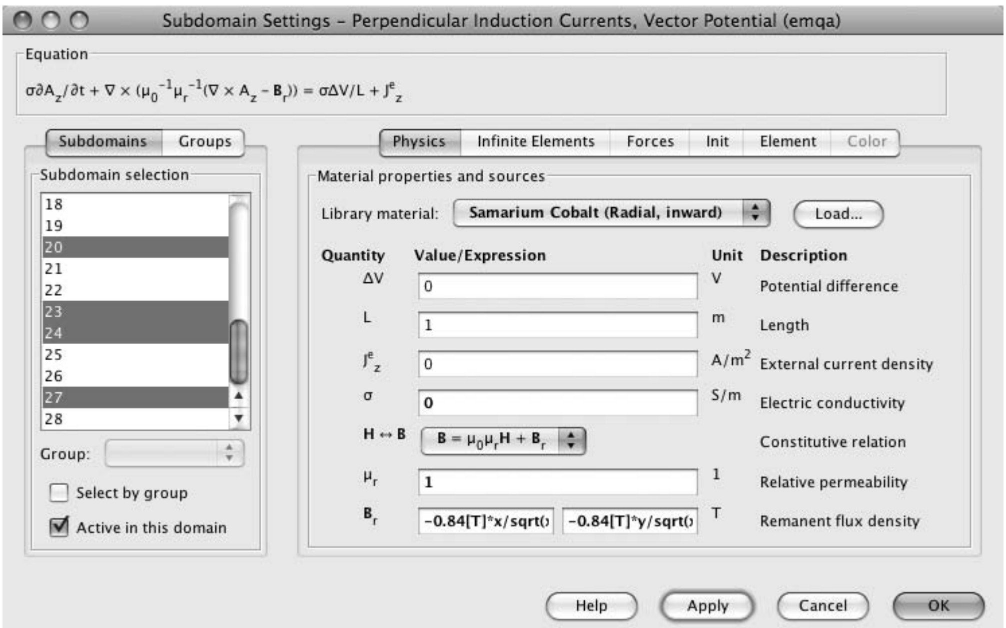


FIGURE 7.81 2D_ACG_1 model Subdomain Settings (20, 23, 24, 27) edit window

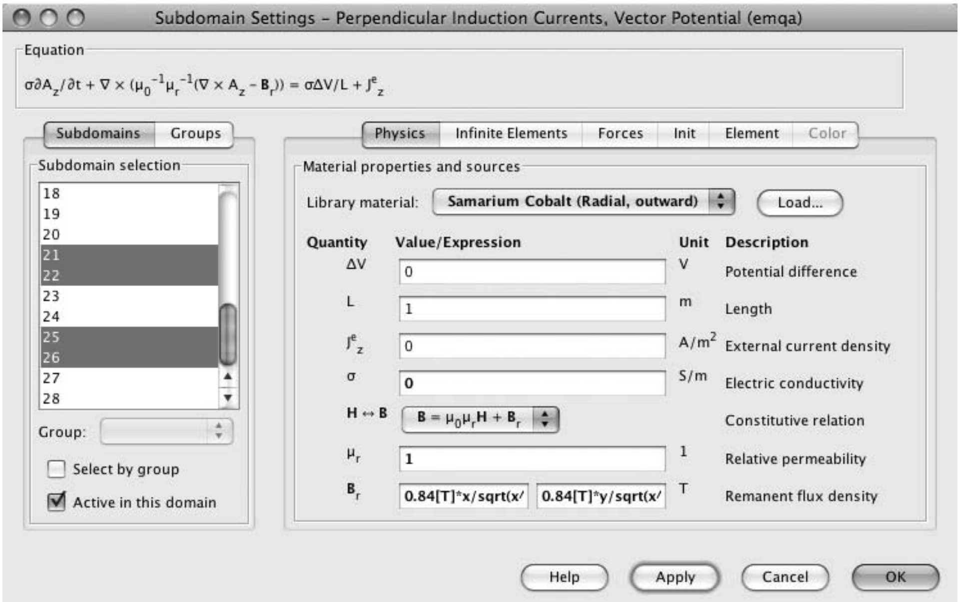


FIGURE 7.82 2D_ACG_1 model Subdomain Settings (21, 22, 25, 26) edit window

Mesh Generation

From the toolbar, select Mesh > Free Mesh Parameters > Global. Select Predefined mesh sizes > Finer (from the pull-down list). Select "Custom mesh size." Enter 2 in the Resolution of narrow regions edit window. See Figure 7.84.

Click the Remesh button, and then click OK. See Figure 7.85.

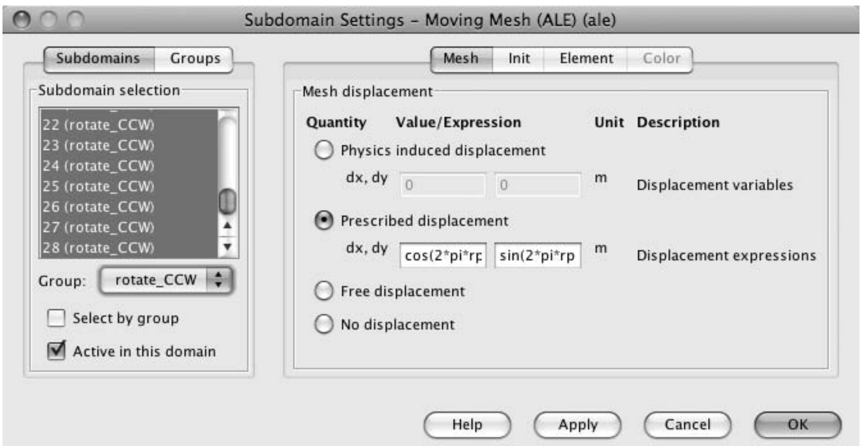


FIGURE 7.83 2D_ACG_1 model Subdomain Settings (19–28) Moving Mesh edit window

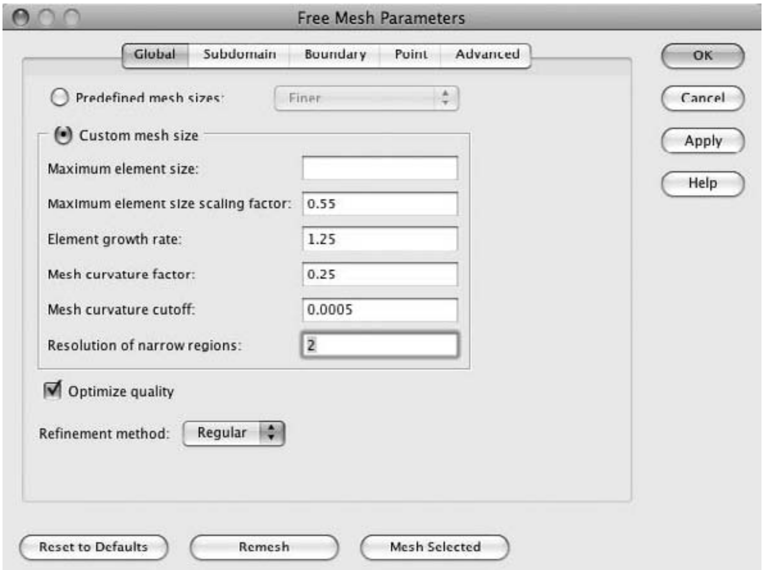


FIGURE 7.84 2D_ACG_1 model Free Mesh Parameters edit window

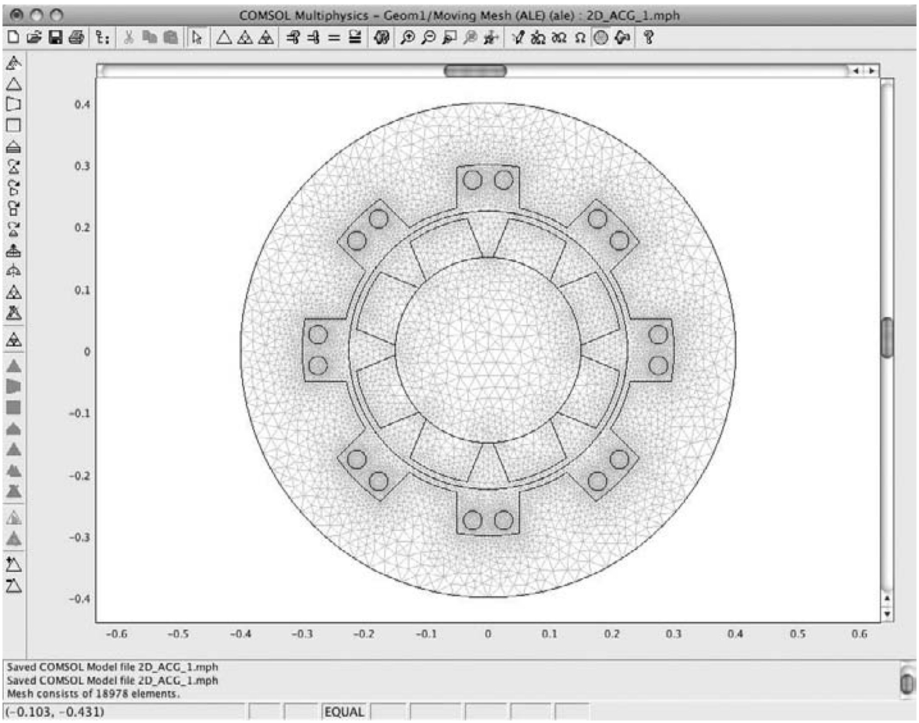


FIGURE 7.85 2D_ACG_1 model mesh

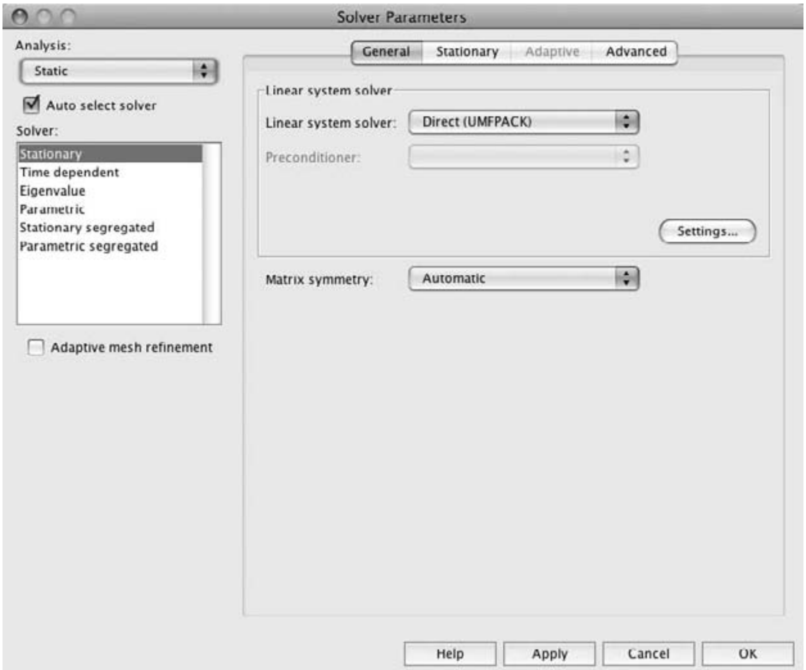


FIGURE 7.86 2D_ACG_1 model Solver Parameters edit window

Solving the Static 2D_ACG_1 Model

Using the menu bar, select Solve > Solver Parameters.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Select "Static" from the Analysis pull-down list. See Figure 7.86. Click OK.

Using the menu bar, select Solve > Solve Problem. See Figure 7.87.

Click the Save button. Select File > Save As. Enter 2D_ACG_2.mph in the Save As edit window. See Figure 7.88. Click OK.

NOTE The 2D_ACG_1 model (static) solution was built to gain experience in the creation of a complex geometrical model. It was saved as 2D_ACG_2.mph and will act as the initial estimate for the 2D_ACG_2 model (transient) solution.

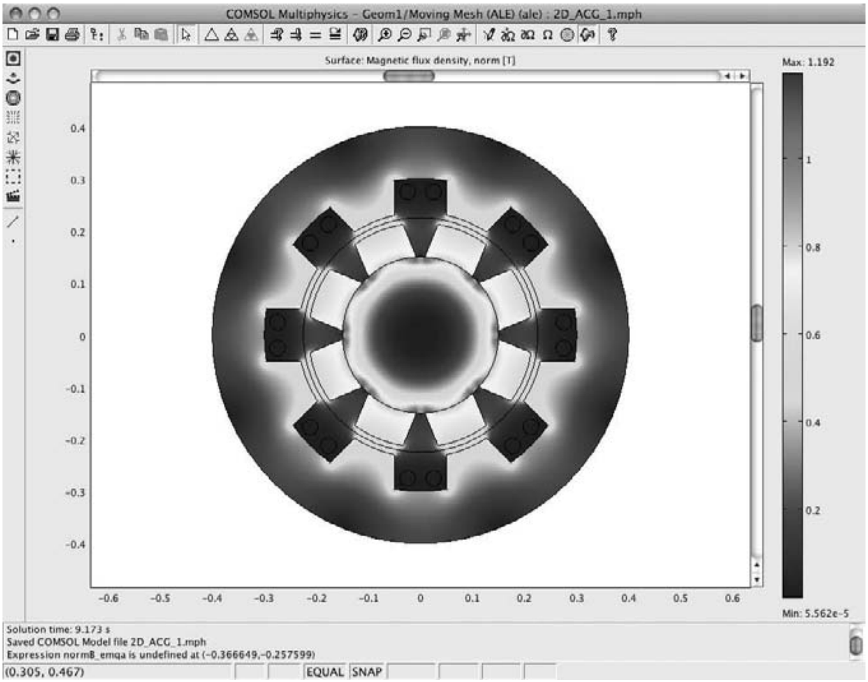


FIGURE 7.87 2D_ACG_1 model solution

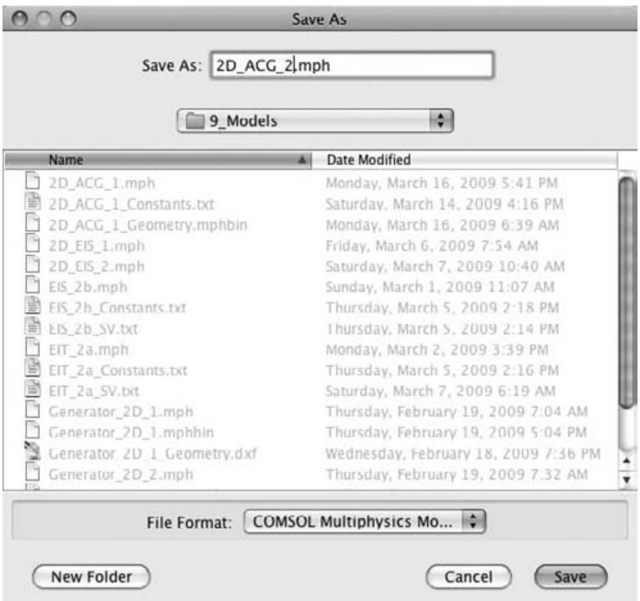


FIGURE 7.88 2D_ACG_2 model initial solution

FIGURE 7.89 2D_ACG_2 Model Navigator initial solution selection

2D AC Generator Model (2D_ACG_2): Transient

The following numerical solution model (2D_ACG_2) follows directly from the earlier model 2D_ACG_1 model, which was built in the preceding subsection. In this next subsection, the transient 2D generator model (2D_ACG_2) uses the static 2D_ACG_1 model as the initial solution to the transient problem. The transient version avoids all the complex geometrical building by starting with the earlier saved solution.

To start building the 2D_ACG_2 model (transient) solution, activate the COMSOL Multiphysics software. In the Model Navigator, select "Open." Select "2D_ACG_2.mph." See Figure 7.89.

Click OK. See Figure 7.90.

NOTE Because the initial solution to the transient 2D_ACG_2 model has already been built and verified, the modeler can proceed directly to implementing the necessary transient solver setup parameters.

Solving the Transient 2D_ACG_2 Model

Using the menu bar, select Solve > Solver Parameters. Select "Transient" from the Analysis pull-down list.

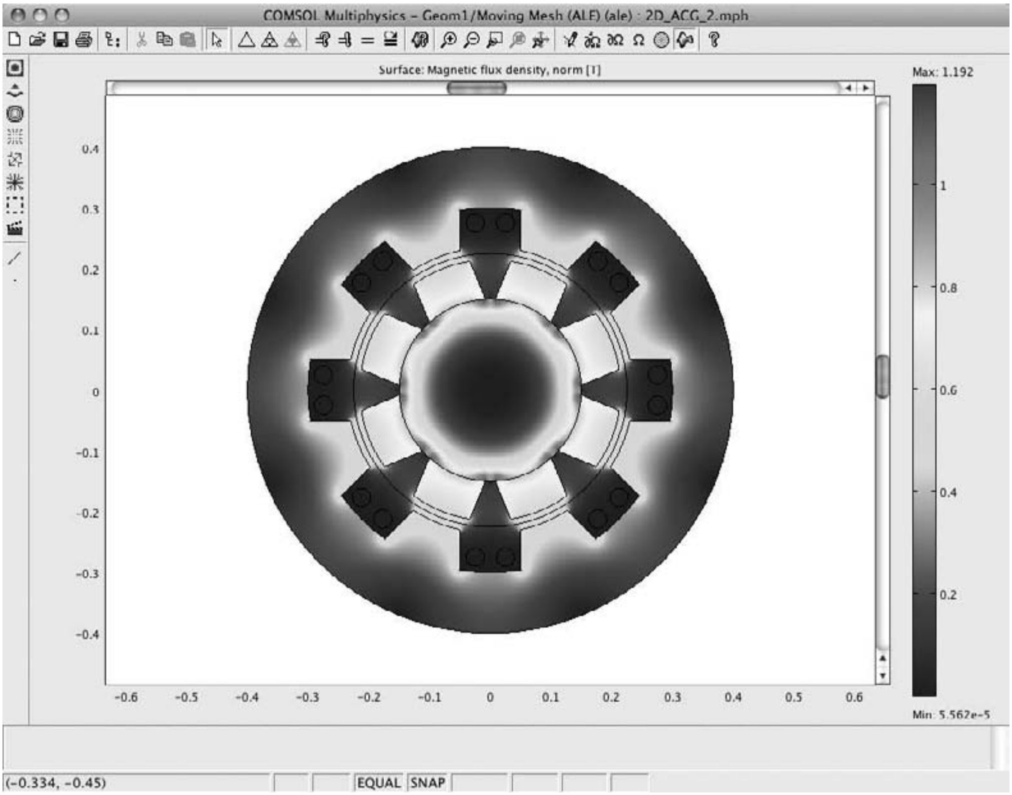


FIGURE 7.90 2D_ACG_2 Model Navigator initial solution

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

In the Times edit window, enter linspace(0,0.25,26). For later versions of COMSOL Multiphysics software enter range(0,0.25/25,0.25). In the Absolute tolerance edit window, enter Az 1e-3 lm1 5e3. Click the Apply button. See Figure 7.91. Click OK.

NOTE It is important to enter the Solver Parameters *exactly* as specified; otherwise, the modeler will see error messages.

Using the menu bar, select Solve > Restart. See Figure 7.92.

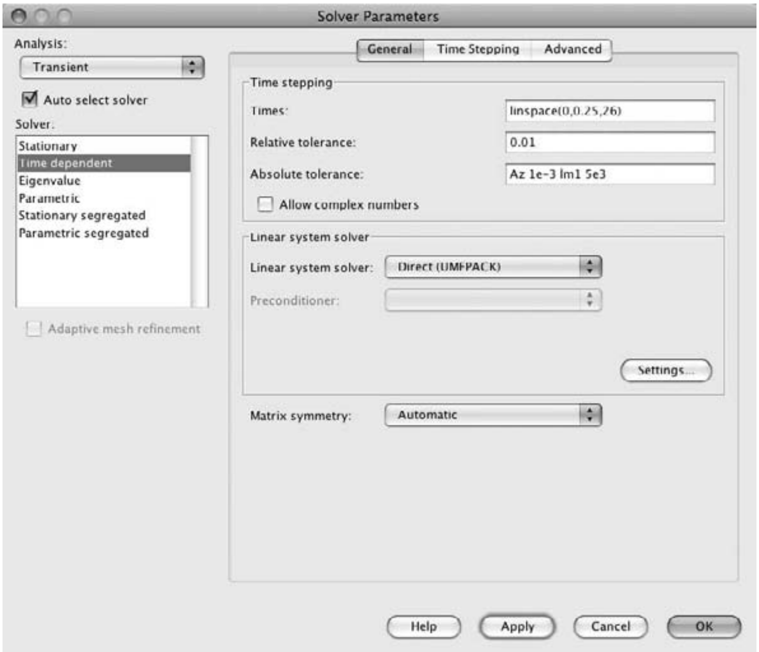


FIGURE 7.91 2D_ACG_2 model Solver Parameters edit window

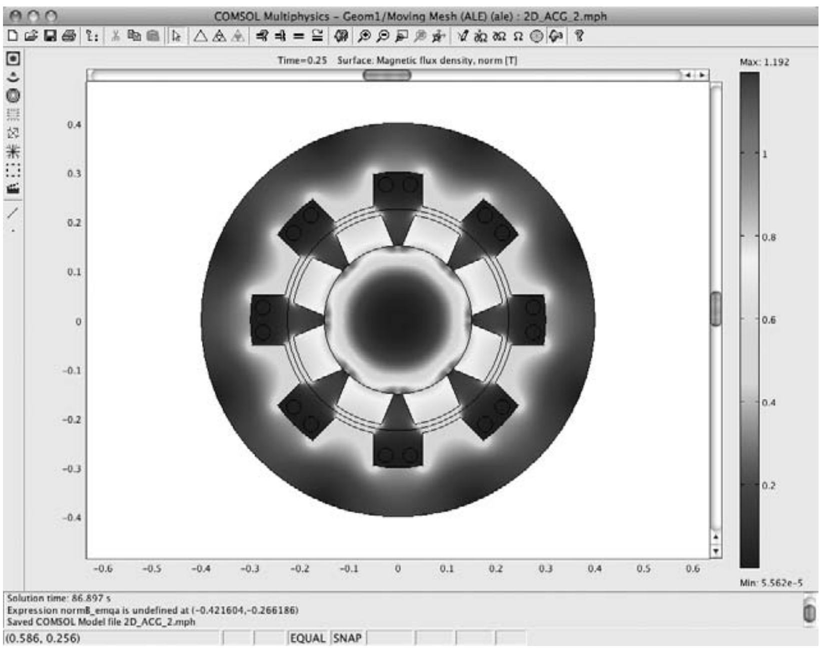


FIGURE 7.92 2D_ACG_2 model solution (transient)

FIGURE 7.93 2D_ACG_2 model Plot Parameters, General tab

Postprocessing and Visualization

The default plot shows a surface plot of the normal magnetic flux density in teslas for the solution at 0.25 second. To visualize the field distribution, the plot parameters will need to be modified.

Select Postprocessing > Plot Parameters > General. Check the Surface and Contour check boxes under Plot type. Uncheck the Geometry edges check box. Select "Solution at time 0.2 seconds" from the Solutions to use pull-down list. See Figure 7.93.

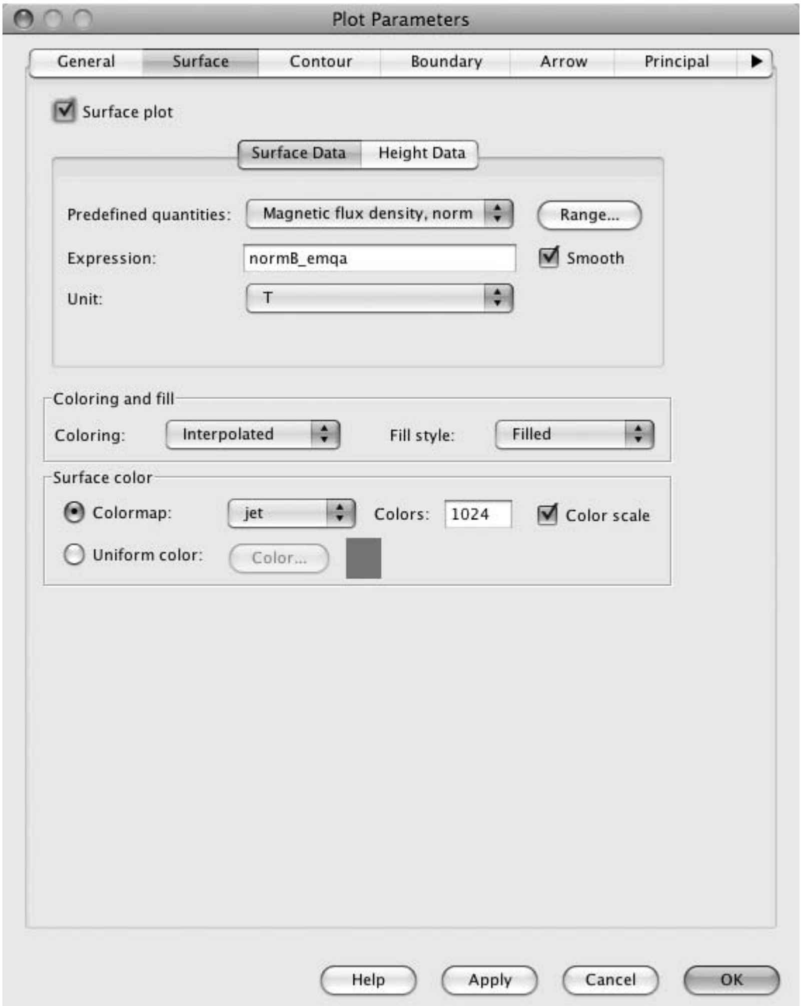


FIGURE 7.94 2D_ACG_2 model Plot Parameters, Surface tab

Click the Surface tab. Select "Magnetic flux density, norm." See Figure 7.94.

Click the Contour tab. Enter Az in the Expression edit window. Enter 15 in the Levels edit window. Click the Uniform color radio button. Click the Color button, and select black. See Figure 7.95.

FIGURE 7.95 2D_ACG_2 model Plot Parameters, Contour tab

Click OK. See Figure 7.96.

To view the induced voltage as a function of time, select Postprocessing > Domain Plot Parameters > Point. Select point 10. Enter Vi*NN in the expression edit window. See Figure 7.97.

Click OK. See Figure 7.98.

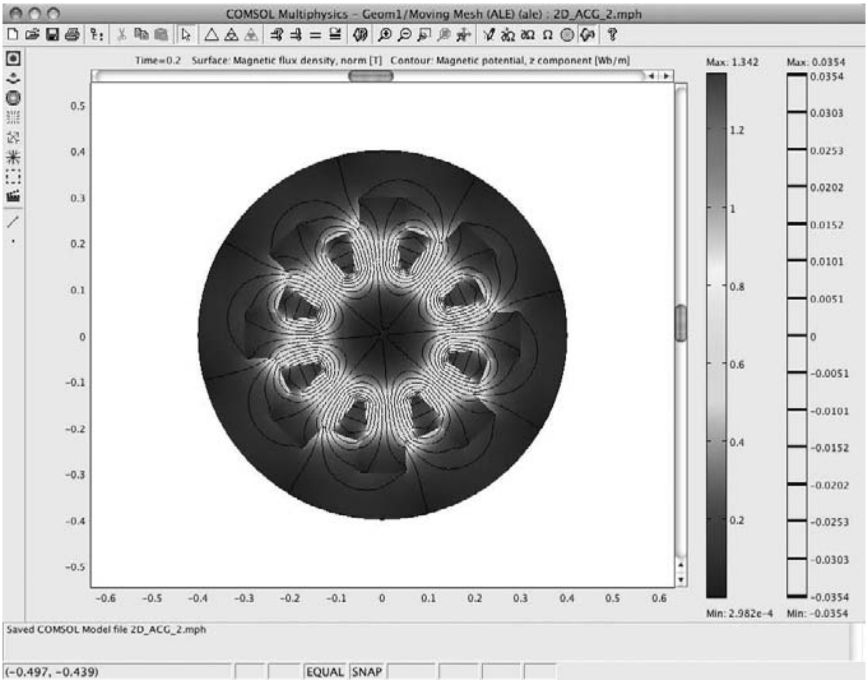


FIGURE 7.96 2D_ACG_2 model magnetic flux density and magnetic potential

FIGURE 7.97 2D_ACG_2 model Domain Plot Parameters edit window

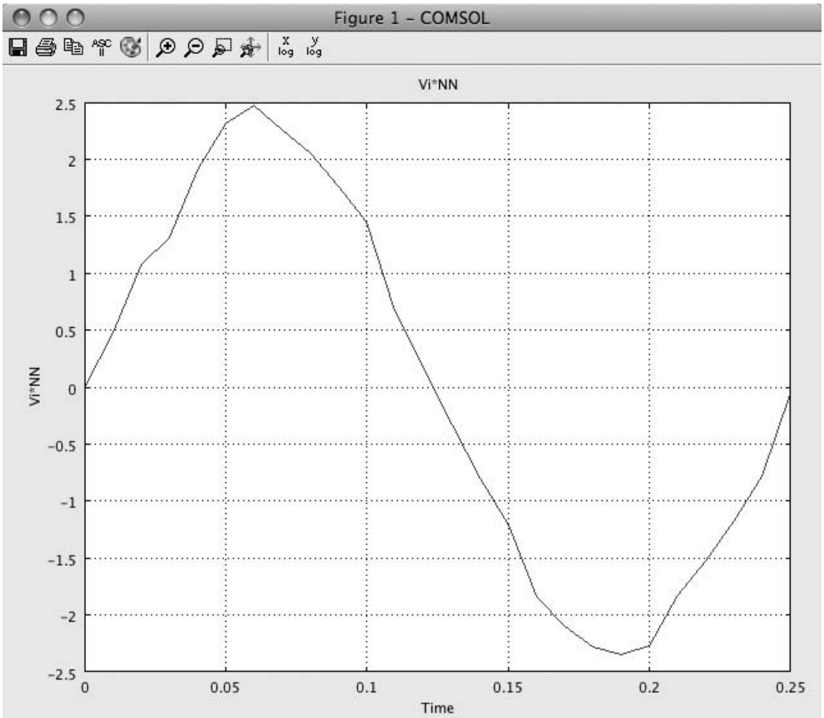


FIGURE 7.98 2D_ACG_2 model induced voltage vs. time

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 7.99.

Click the Start Animation button. See Figure 7.100.

Alternatively, you can play the file Movie7_ACG_2.avi that was supplied with this book.

2D AC Generators, Static and Transient Models: Summary and Conclusions

In this section, we built two 2D AC generator models: static and transient. These models generate low-frequency (60 Hz) current and voltage, as would be typically found on the power transmission grid. They demonstrate the use of both hard (not easily magnetized) and soft (easily magnetized) nonlinear magnetic materials in the construction of a rotating machine for the conversion of mechanical energy to electrical energy.

The modeling and physics concepts employed in this section of Chapter 7 are mechanical to electrical energy conversion, hard and soft nonlinear magnetic materials, moving mesh (ALE), and geometric assembly (pair creation across a boundary).

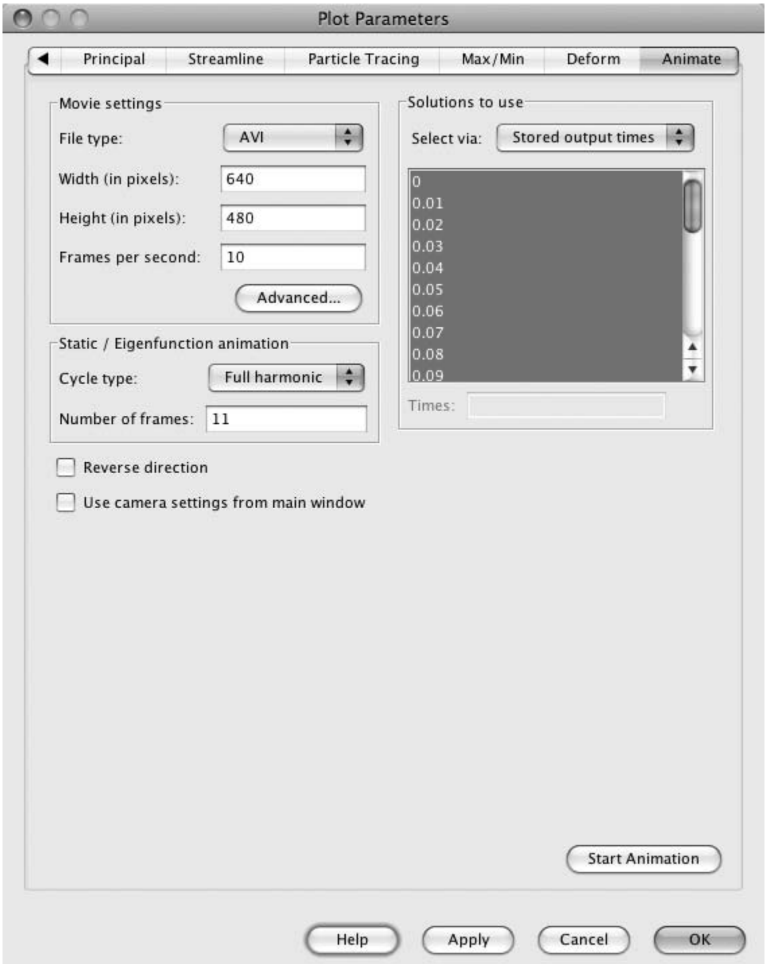


FIGURE 7.99 2D_ACG_2 model Plot Parameters, Animate tab

2D AC Generator: Sector—Static and Transient

In this section, static and transient sector-based models are developed that are only one-eighth the size of the full-size model. The sector models utilize the inherent basic symmetry employed in AC generators. Additionally, an ordinary differential equation[28] is incorporated into the model to allow the exploration of the torques resulting from the magnet forces inherent in this AC generator design. In these machines, the generation of AC power is accomplished through the application of Faraday's law of induction. In the following models, a magnetic vector potential (**A**) is employed that has only the *z* component.

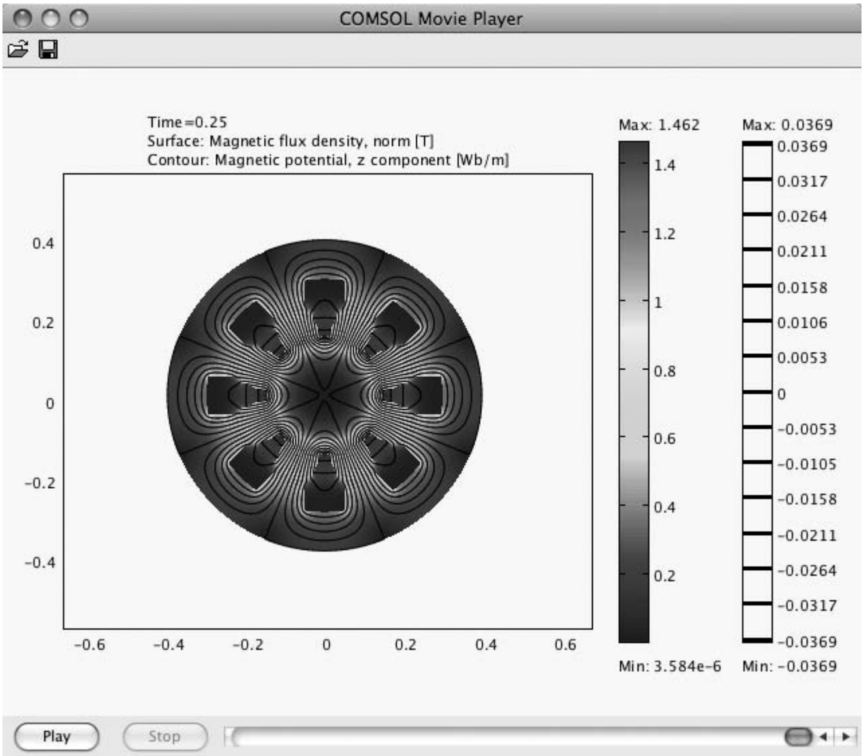


FIGURE 7.100 2D_ACG_2 model animation, final frame

Rotation is modeled using deformed Mesh Application Mode (ALE). The rotor and the stator are drawn separately and then combined as an assembly.

The materials employed in this model are high-energy samarium–cobalt magnets with nonlinear soft iron pole pieces.

2D AC Generator Sector Model (2D_ACGS_1): Static

The following numerical solution model (2D_ACGS_1) is derived from a model that was originally developed by COMSOL as an AC/DC Module Motors and Drives Library Model. First, this 2D generator model will be built as a static (stationary) model. In the next subsection, the static model will be used as the starting point for the transient (rotating) model (2D_ACGS_2).

To start building the 2D_ACGS_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from the Space dimension pull-down list. Select AC/DC Module > Rotating Machinery > Rotating Perpendicular Currents. Enter Az X Y Z in the Dependent variables edit window. See Figure 7.101. Click OK.

FIGURE 7.101 2D_ACGS_1 Model Navigator setup

NOTE Be sure to change the dependent variables from lowercase x, y, z to uppercase X, Y, Z. This demonstrates that the modeler can alter the Reference Frame in COMSOL Multiphysics Software, if needed. Also note that the Model Navigator shows two names in the Application mode name edit window: emqa and ale. Those names—emqa (Rotating Perpendicular Currents) and ale (Moving Mesh)—indicate the Application Modes employed in this modeling analysis.

Select File > Save As. Enter 2D_ACGS_1.mph in the Save As edit window. Click the Save button.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 7.19; also see Figure 7.102. Click OK.

Generator Sector Geometry

Select File > Import > CAD Data From File > 2D_ACG_1_Geometry.mphbin. See Figure 7.103.

Table 7.19 Constants Edit Window

Name	Expression	Description
A	pi*(0.02[m])^2	Area of wires in stator
L	0.4[m]	Length of generator
NN	1	Number of turns in stator winding
M	1400[N*m]	Externally applied torque
Rc	1e-4[ohm]	Resistance of winding
I0	100[kg*m^2]	External moment of inertia

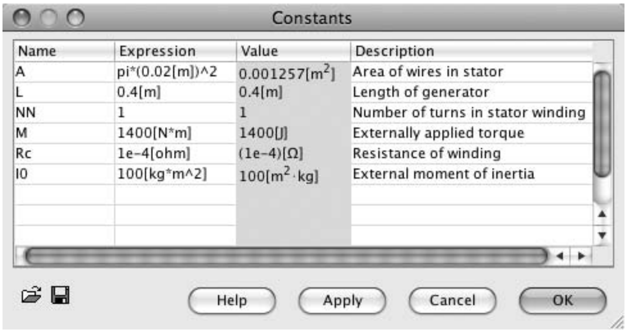


FIGURE 7.102 2D_ACGS_1 model Constants edit window

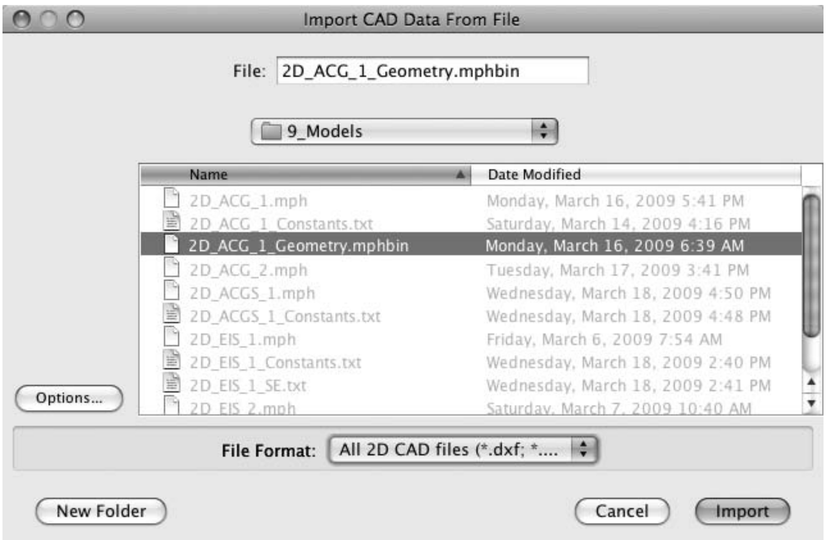


FIGURE 7.103 Import CAD Data From File select window

FIGURE 7.104 Imported CAD file

Click the Import button. See Figure 7.104.

NOTE Now that the geometry for the entire 2D generator has been imported, the modeler needs to create the 2D generator sector. Carefully enter *exactly* the following construction, to create the 2D generator sector.

Using the menu bar, select Draw > Specify Objects > Line. Select "Closed polyline (solid)" from the Style pull-down list. Enter the following formula in the x edit window:

0 0.4*cos(67.5*pi/180) 0.4 0.4*cos(22.5*pi/180) 0

Enter the following formula in the y edit window:

0 0.4*sin(67.5*pi/180) 0.4 0.4*sin(22.5*pi/180) 0

See Figure 7.105.

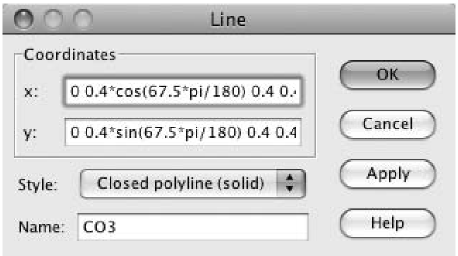


FIGURE 7.105 Line, Closed polyline (solid) edit window

Click OK. See Figure 7.106.

Select the newly created solid (CO3) only. Using the menu bar, select Edit > Copy. Using the menu bar, select Draw > Create Composite Object. Enter CO1*CO3 in the Set formula window. See Figure 7.107. Click OK.

Using the menu bar, select Edit > Paste. Click OK on the x = 0, y = 0 Displacements dialog box.

FIGURE 7.106 2D generator geometry and closed polyline (solid)

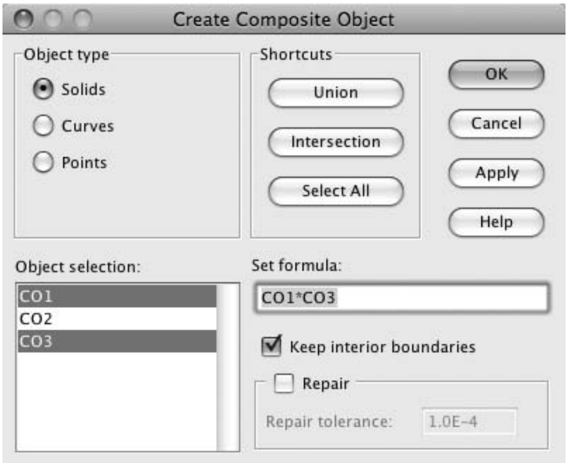


FIGURE 7.107 Create Composite Object, intersection (CO1, CO3)

Using the menu bar, select Draw > Create Composite Object. Enter CO2*CO1 in the Set formula window. See Figure 7.108.

Click OK, and then click the Zoom Extents button. See Figure 7.109.

Assemble the Geometry (Stator and Rotor)

Select both objects (CO3, CO4). Locate the Create Pairs button on the Draw toolbar. See Figure 7.110.

Click the Create Pairs button on the Draw toolbar. See Figure 7.111.

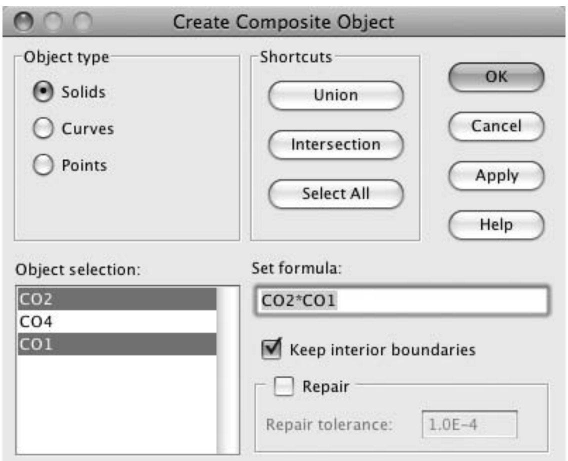


FIGURE 7.108 Create Composite Object, intersection (CO2, CO1)

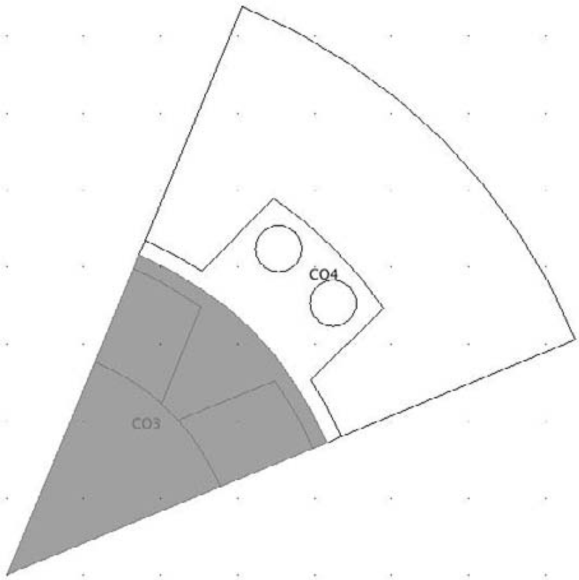


FIGURE 7.109 Created one-eighth generator sector (CO3, CO4)

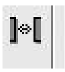


FIGURE 7.110 Create Pairs button on Draw toolbar

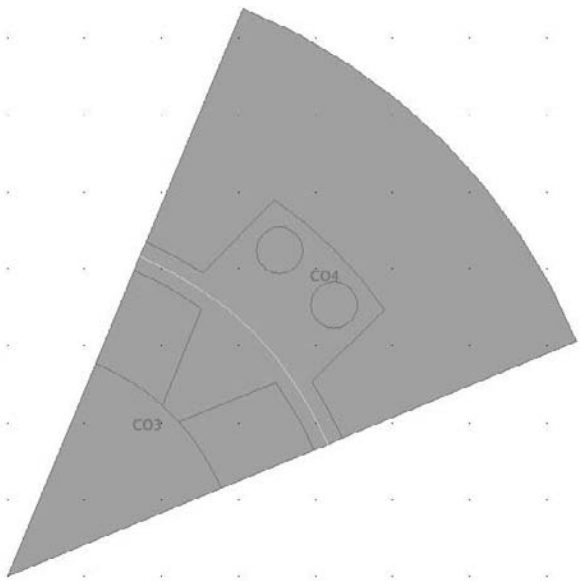


FIGURE 7.111 2D_ACGS_1 model paired stator (CO3) and rotor (CO4)

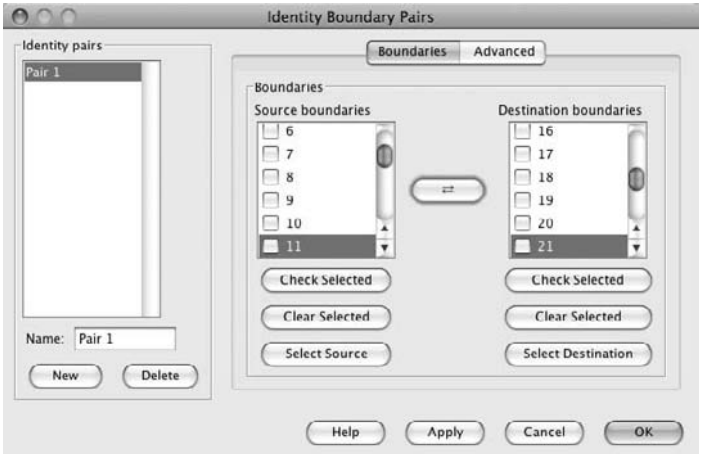


FIGURE 7.112 Identity Boundary Pairs window

NOTE The pairing of the rotor and the stator couples the boundaries of the two separately created geometric elements. This pairing facilitates the use of the sliding mesh at the boundary between the rotor and the stator, which would otherwise be discontinuous.

Using the menu bar, select Physics > Identity Pairs > Identity Boundary Pairs. Select Pair 1. See Figure 7.112.

Click the Interchange source and destination button, which is located between the Source boundaries and Destination boundaries fields. See Figure 7.113. Click OK.

Options: Global Expressions

Using the menu bar, select Options > Expressions > Global Expressions. In the Global Expressions edit window, enter the information shown in Table 7.20; also see Figure 7.114. Click OK.

FIGURE 7.113 Interchange source and destination button

Table 7.20 Global Expressions Edit Window

Name	Expression	Description
Tz	8*L[1/m]*F_torquez_emqa	Total torque for the entire device

FIGURE 7.114 2D_ACGS_1 model Global Expressions edit window

NOTE The insertion of [1/m] in the expression for Tz converts L into a unitless number. Because of the use of coupling variables, COMSOL Multiphysics software cannot determine the units of other variables that depend either directly or indirectly on the coupling variables. This may cause inconsistent unit warnings to appear. Such warnings can be ignored.

Options: Subdomain Expressions

Using the menu bar, select Options > Expressions > Subdomain Expressions. In the Subdomain Expressions edit window, enter the information shown in Table 7.21. Click OK. See Figures 7.115 and 7.116.

Using the menu bar, select Options > Expressions > Subdomain Expressions. In the Subdomain Expressions edit window, enter the information shown in Table 7.22. Click OK. See Figures 7.117 and 7.118.

Options: Subdomain Integration Variables

Using the menu bar, select Options > Integration Coupling Variables > Subdomain Variables. In the Subdomain Integration Variables edit window, enter the information shown in Table 7.23. Click OK. See Figures 7.119 and 7.120.

Table 7.21 Subdomain Expressions Edit Window

Subdomain	Name	Expression	Figure Number
1–4	u	−y*alphat	7.115
1–4	v	x*alphat	7.115
5–8	u	0	7.116
5–8	v	0	7.116

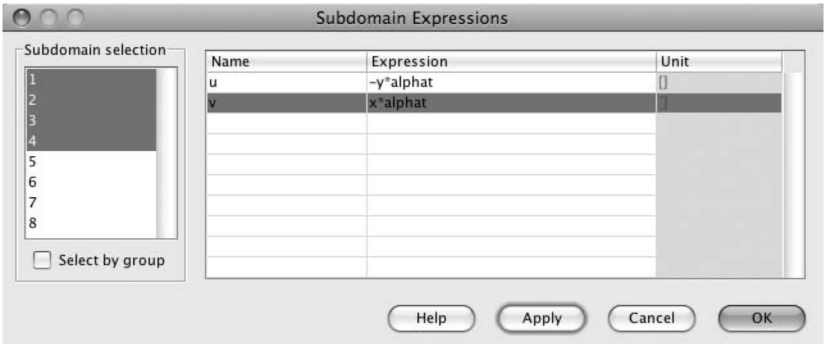


FIGURE 7.115 2D_ACGS_1 model Subdomain Expressions (1–4) edit window

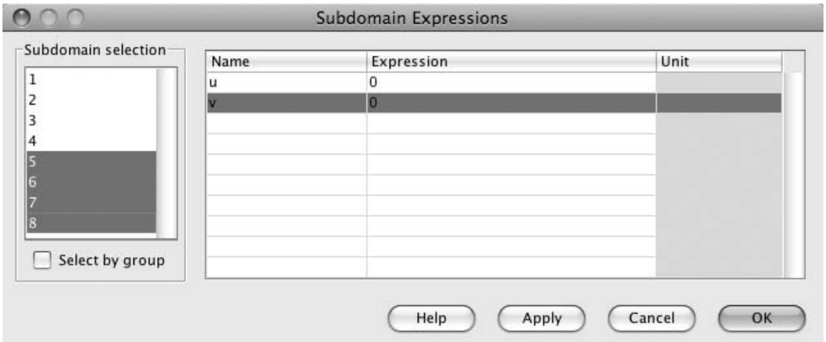


FIGURE 7.116 2D_ACGS_1 model Subdomain Expressions (5–8) edit window

Table 7.22 Subdomain Expressions Edit Window

Subdomain	Name	Expression	Figure Number
1	rho	7870[kg/m^3]	7.117
2, 4	rho	8400[kg/m^3]	7.118

Table 7.23 Subdomain Integration Variables Edit Window

Subdomain	Name	Expression	Integration Order	Global	Figure Number
1, 2, 4	I	8*L*rho*(x^2+y^2)	4	Yes	7.119
7, 8	Vi	8*L*NN*Ez_emqa/A	4	Yes	7.120

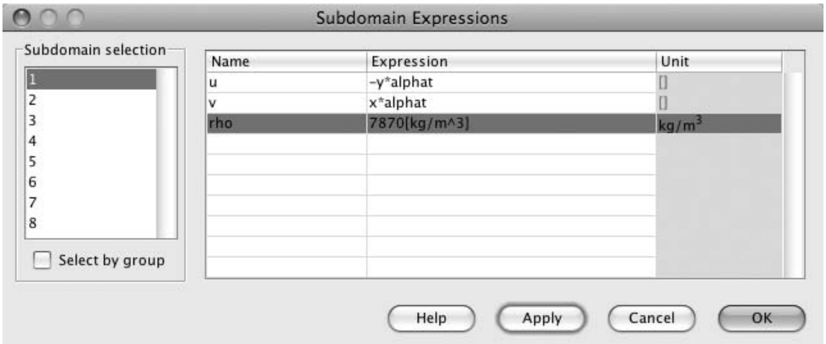


FIGURE 7.117 2D_ACGS_1 model Subdomain Expressions (1) edit window

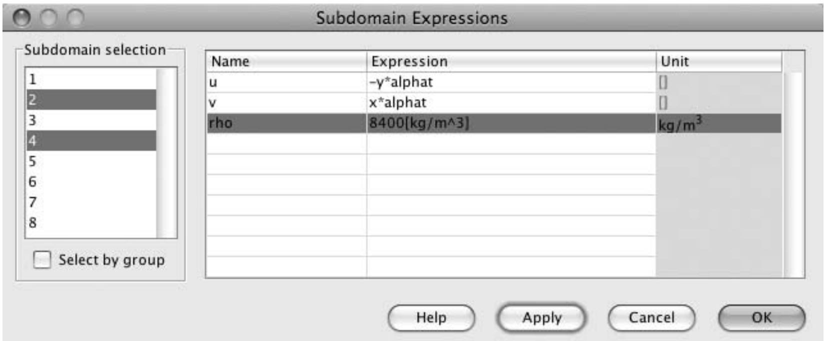


FIGURE 7.118 2D_ACGS_1 model Subdomain Expressions (2, 4) edit window

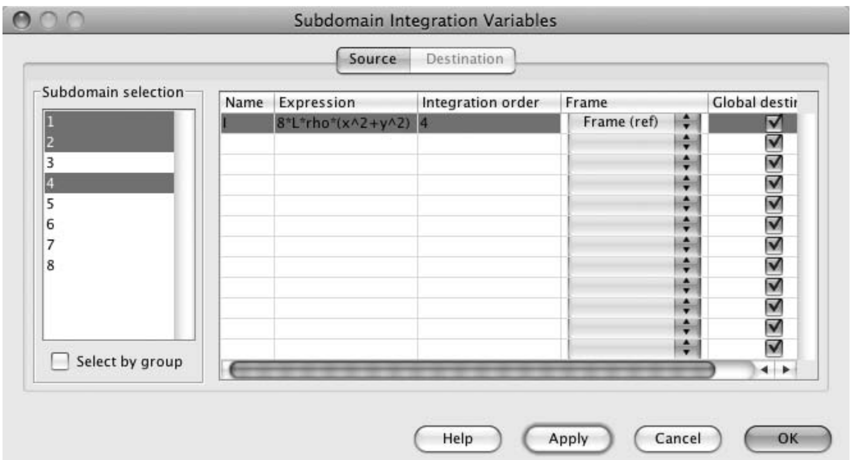


FIGURE 7.119 2D_ACGS_1 model Subdomain Integration Variables (1, 2, 4) edit window

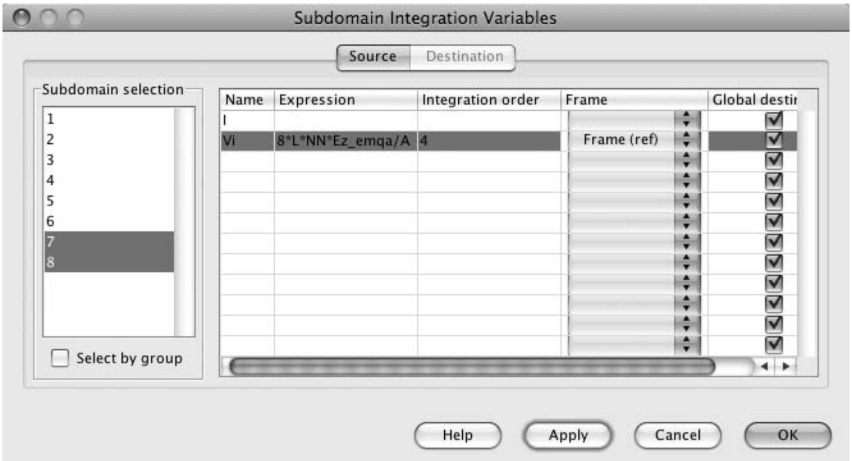


FIGURE 7.120 2D_ACGS_1 model Subdomain Integration Variables (7, 8) edit window

---

NOTE Because of the use of coupling variables, COMSOL Multiphysics software cannot determine the units of other variables that depend either directly or indirectly on the coupling variables. This may cause inconsistent unit warnings to appear. Such warnings can be ignored.

Physics Subdomain Settings: Moving Mesh (ALE) (ale)

Using the menu bar, select Multiphysics > Moving Mesh (ALE) (ale). Select Physics > Subdomain Settings. Select subdomains 1–4. Select "angle_CCW" from the Group pull-down list. See Figure 7.121. Click OK.

NOTE The selection of "angle_CCW" makes the variable omega available for use in the ODE, once defined later.

Application Mode Properties: Perpendicular Induction Currents, Vector Potential (emqa)

Using the menu bar, select Multiphysics > Perpendicular Induction Currents, Vector Potential (emqa). Select Physics > Properties. Select "Frame (ref)" from the Frame pull-down list. Select "On" from the Weak constraints pull-down list. See Figure 7.122. Click OK.

Physics Subdomain Settings: Perpendicular Induction Currents, Vector Potential (emqa)

Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window. Click the Load button.

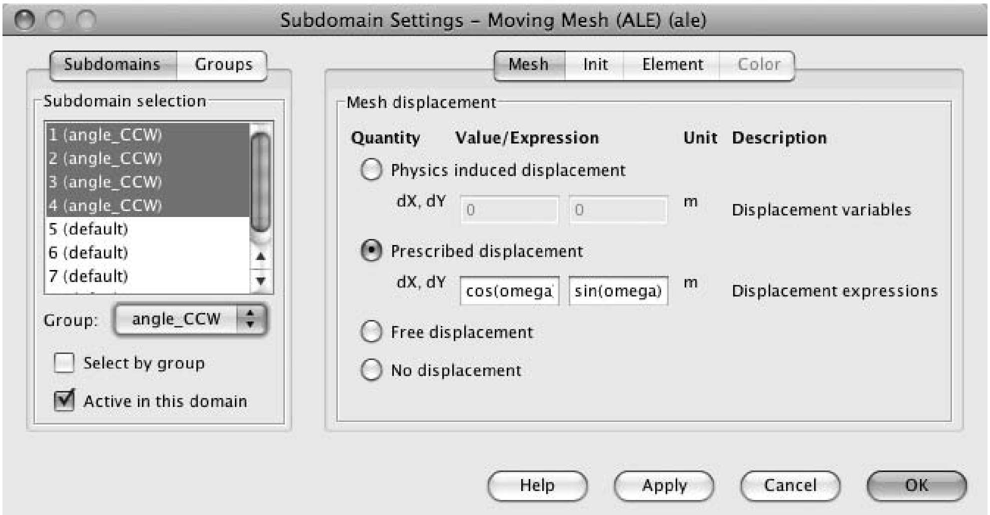


FIGURE 7.121 2D_ACGS_1 model Subdomain Settings (1–4) Moving Mesh (ALE) edit window

Select Electric (AC/DC) Materials Properties Library > Soft Iron (without losses). Click the Apply button.

Select Electric (AC/DC) Materials Properties Library > Samarium Cobalt (Radial, inward). Click the Apply button.

Select Electric (AC/DC) Materials Properties Library > Samarium Cobalt (Radial, outward). Click the Apply button. See Figure 7.123. Click the Cancel button.

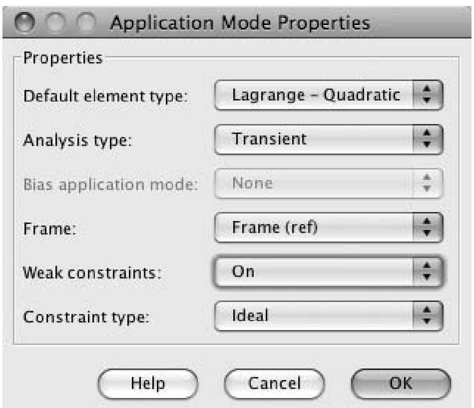


FIGURE 7.122 2D_ACGS_1 model Application Mode Properties, Perpendicular Induction Currents, Vector Potential (emqa)

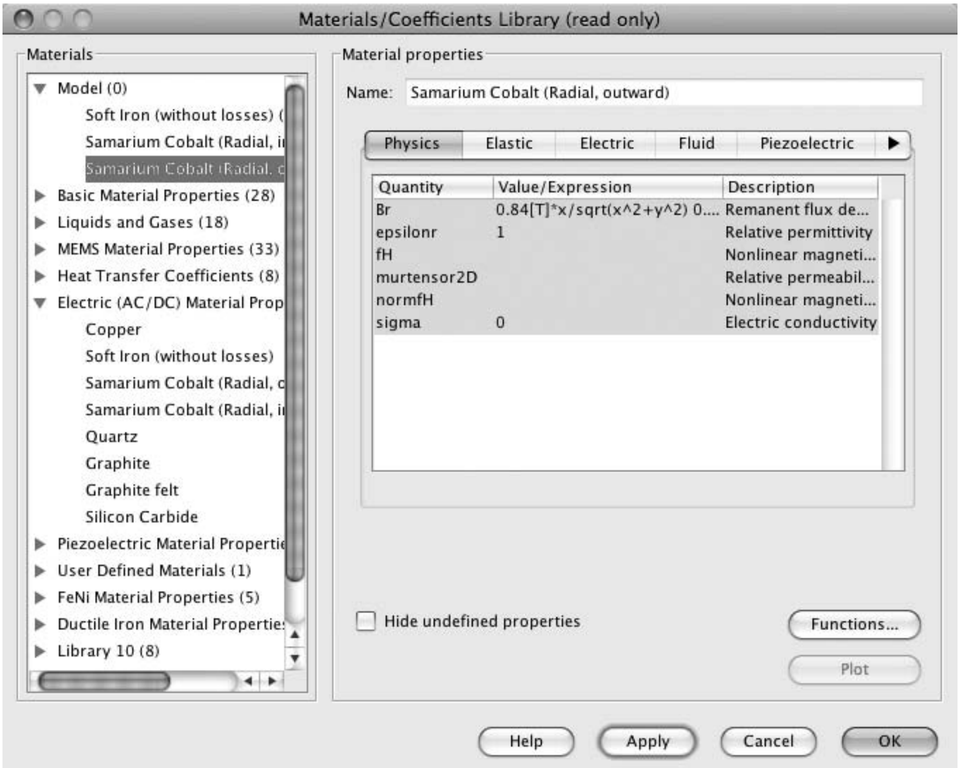


FIGURE 7.123 2D_ACGS_1 model Materials/Coefficients Library load window

NOTE The last three commands added the three selected materials to the Model (0) Library for use in this model.

In the Subdomain edit windows, enter the information shown in Table 7.24. Click the Apply button. See Figures 7.124–7.127.

Table 7.24 Subdomain Edit Windows

Subdomain	H ⇔ B Setting	Material	Figure Number
1, 6	$H = f(\|B\|)e_B$	Soft iron (without losses)	7.124
2	$B = \mu_0\mu_r H + B_r$	Samarium cobalt (radial, outward)	7.125
4	$B = \mu_0\mu_r H + B_r$	Samarium cobalt (radial, inward)	7.126
3, 5, 7, 8	$B = \mu_0\mu_r H$	—	7.127

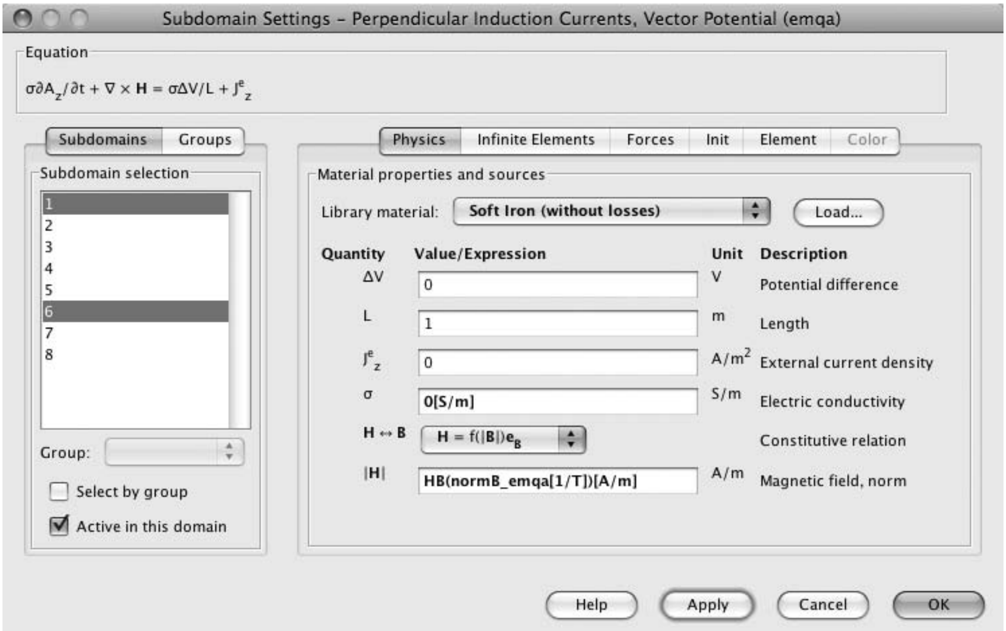


FIGURE 1.124 2D_ACGS_1 model Subdomain Settings (1, 6) edit window

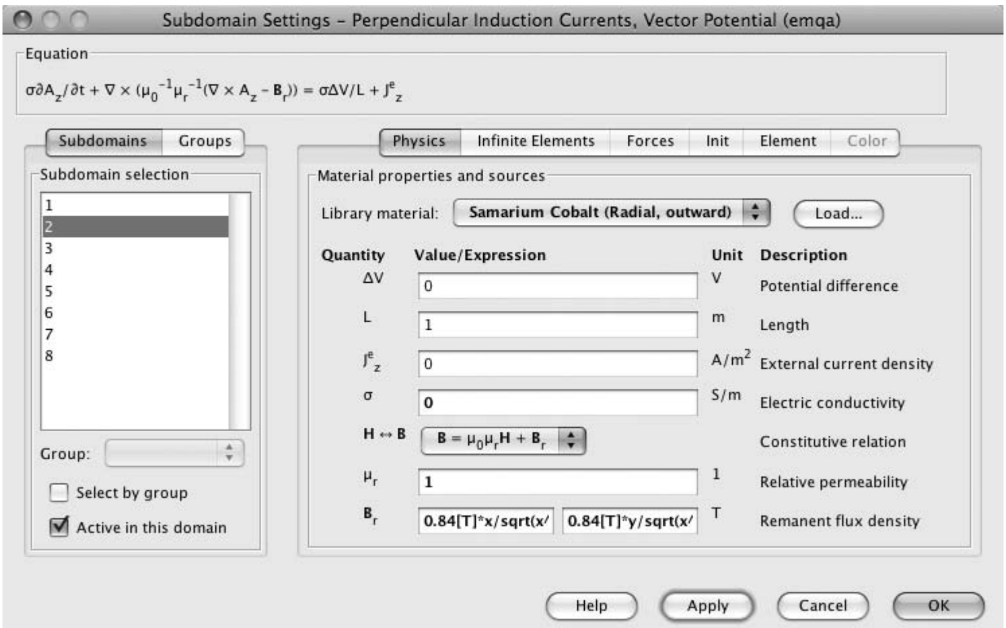


FIGURE 7.125 2D_ACGS_1 model Subdomain Settings (2) edit window

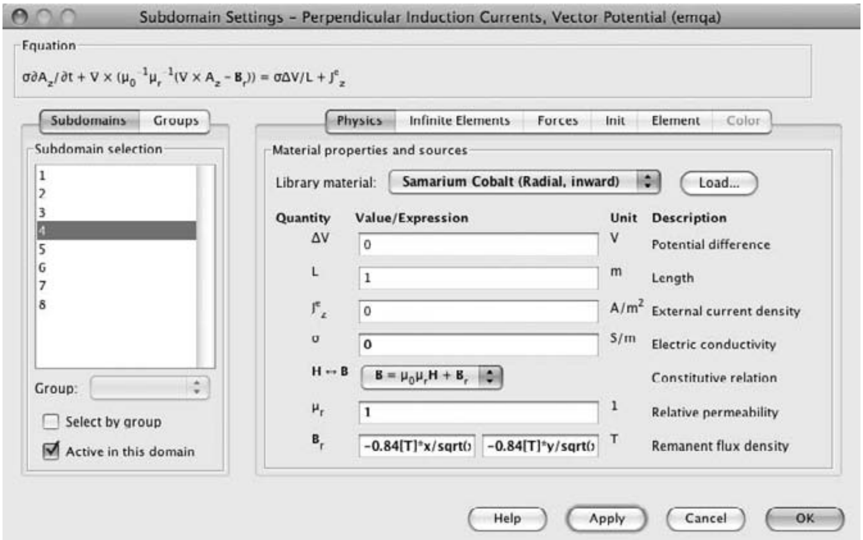


FIGURE 7.126 2D_ACGS_1 model Subdomain Settings (4) edit window

Select subdomains 7 and 8. Enter NN*Vi/(Rc*A) in the External current density edit window. See Figure 7.128. Click the Apply button.

Click the Forces tab. Select subdomains 1, 2, and 4. Enter F as the name of the variable. Click the Apply button. See Figure 7.129. Click OK.

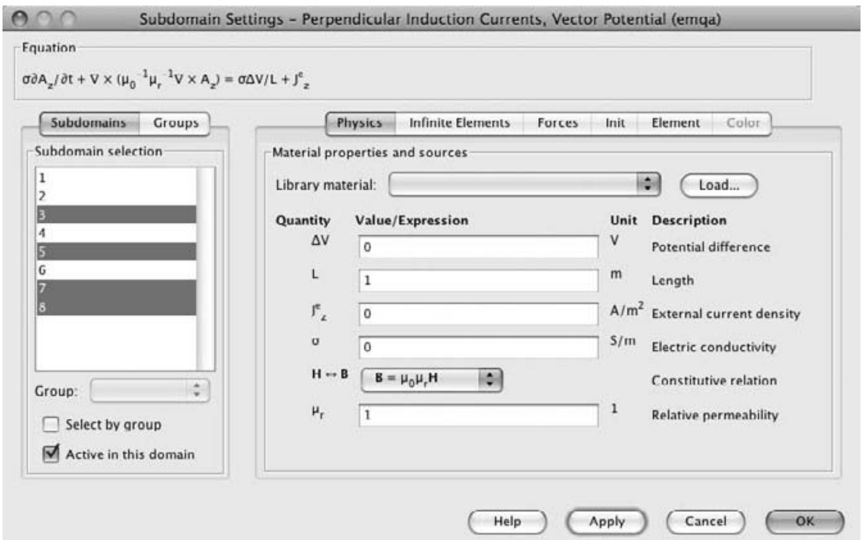


FIGURE 7.127 2D_ACGS_1 model Subdomain Settings (3, 5, 7, 8) edit window

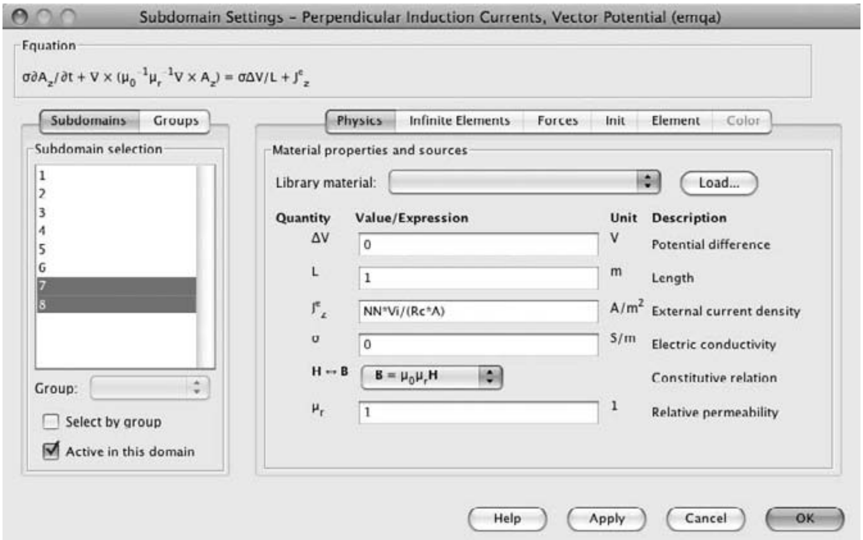


FIGURE 7.128 2D_ACGS_1 model Subdomain Settings (7, 8) edit window

Physics Boundary Settings: Perpendicular Induction Currents, Vector Potential (emqa)

Using the menu bar, Select Physics > Boundary Settings. Select boundaries 1–4, 7, 8, 15, 16, 19, and 20 in the Boundary selection window.

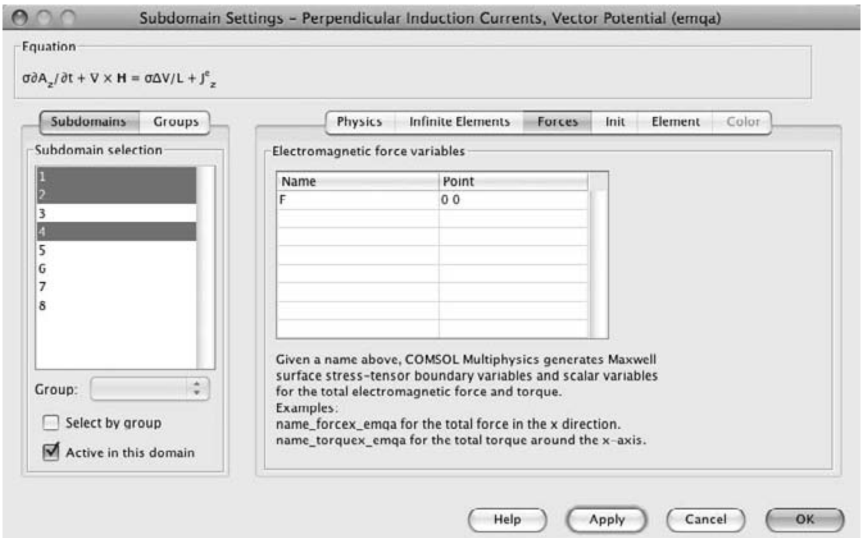


FIGURE 7.129 2D_ACGS_1 model Subdomain Settings (1, 2, 4) Forces edit window

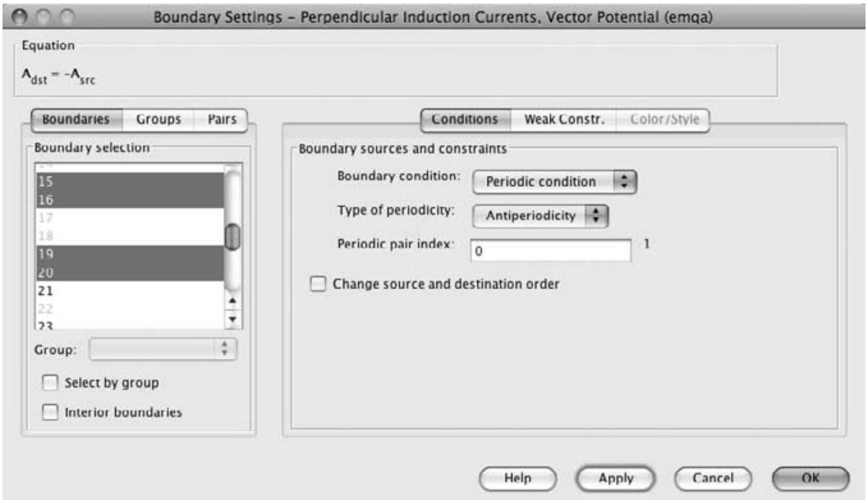


FIGURE 7.130 2D_ACGS_1 model Boundary Settings (1–4, 7, 8, 15, 16, 19, 20) Conditions page

On the Conditions page, select "Periodic condition" from the Boundary condition pull-down list. Also select "Antiperiodicity" from the Type of periodicity pull-down list. See Figure 7.130.

Click the Weak Constr. tab. Select boundaries 1–4, 7, 8, 15, 16, 19–21, and 23 in the Boundary selection window. Uncheck the Use weak constraints check box. Click the Apply button. See Figure 7.131.

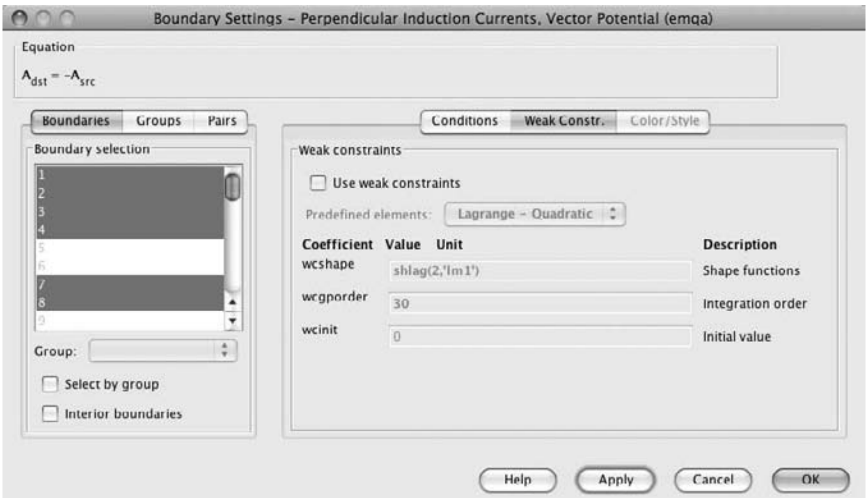


FIGURE 7.131 2D_ACGS_1 model Boundary Settings (1–4, 7, 8, 15, 16, 19–21, 23) Weak Constr. page

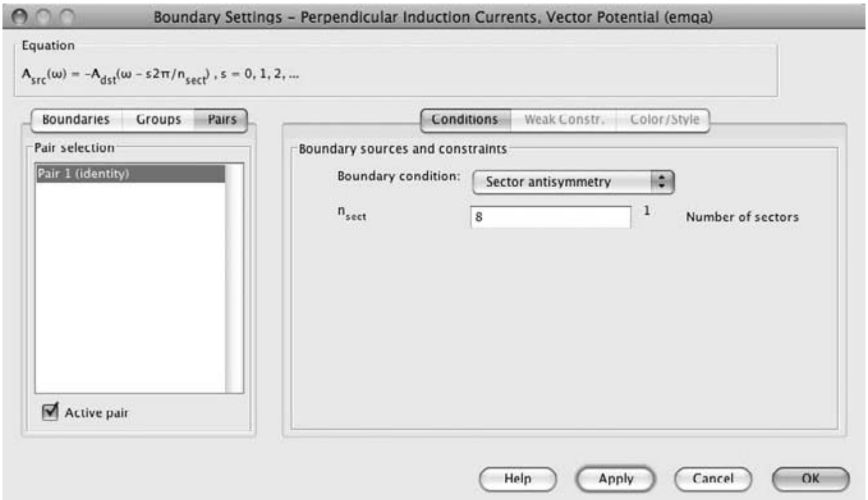


FIGURE 7.132 2D_ACGS_1 model Boundary Settings Pairs selection (1)

Click the Pairs tab. Select Pair 1. Select "Sector antisymmetry" from the Boundary condition pull-down list. Enter 8 in the Number of sectors edit window. See Figure 7.132. Click OK.

Physics Settings: Periodic Point Conditions

Using the menu bar, select Physics > Periodic Conditions > Periodic Point Conditions. Select point 4 in the Point selection window. On the Source page, enter lm1 (Lagrange Multiplier 1) in the first row of the Expression edit window. Press Return. See Figure 7.133.

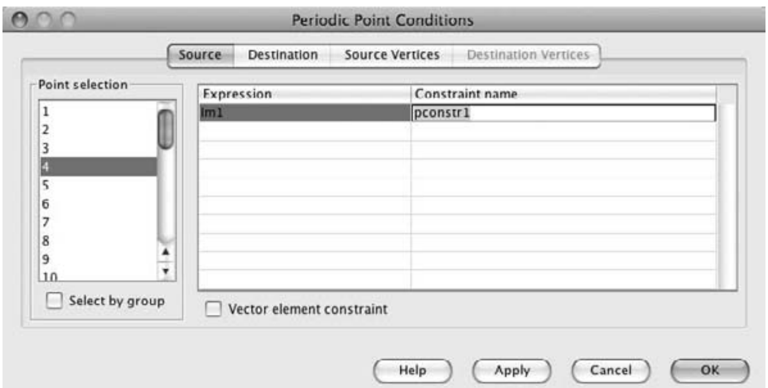


FIGURE 7.133 2D_ACGS_1 model Periodic Point Conditions (4) Source page

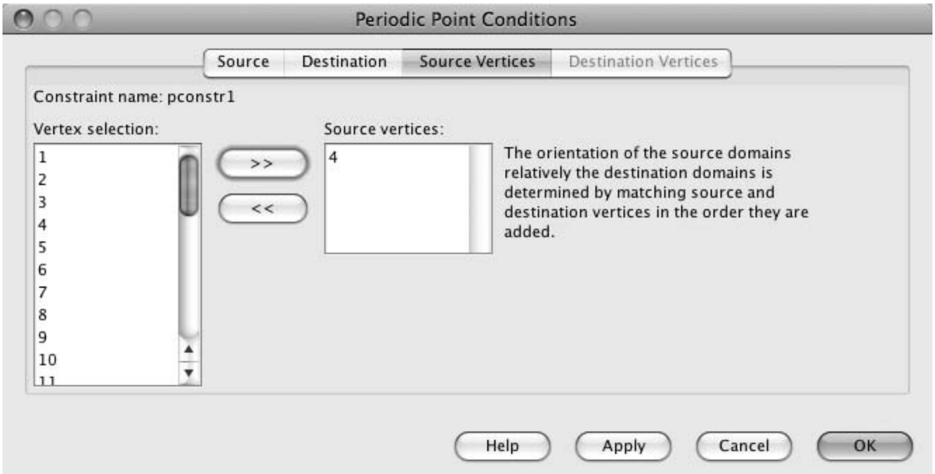


FIGURE 7.134 2D_ACGS_1 model Periodic Point Conditions, Source Vertices (4) page

---

NOTE When the modeler presses the Return key, pconstr1 appears in the Constraint name window.

Click the Source Vertices tab. Click the >> button to select point 4 as a source vertex. See Figure 7.134.

Click the Destination tab. Select point 11. Check the Use selected points as destination check box. Enter -lm1. See Figure 7.135.

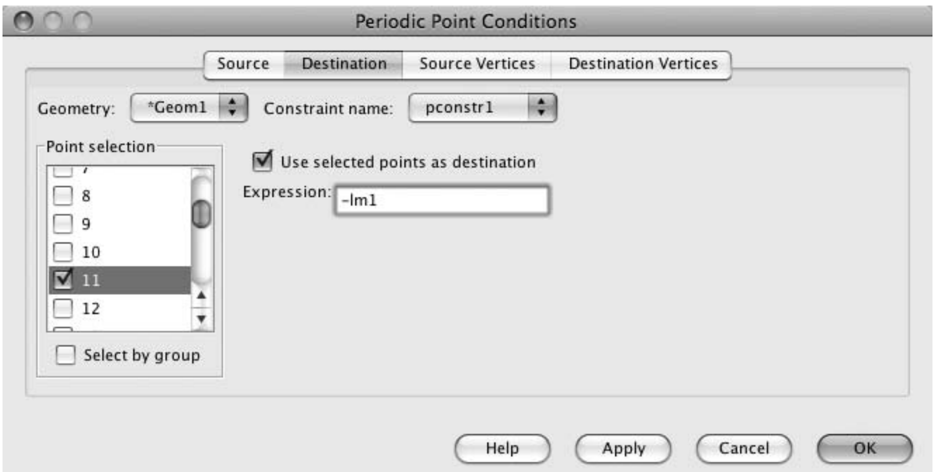


FIGURE 7.135 2D_ACGS_1 model Periodic Point Conditions, Destination page (11)

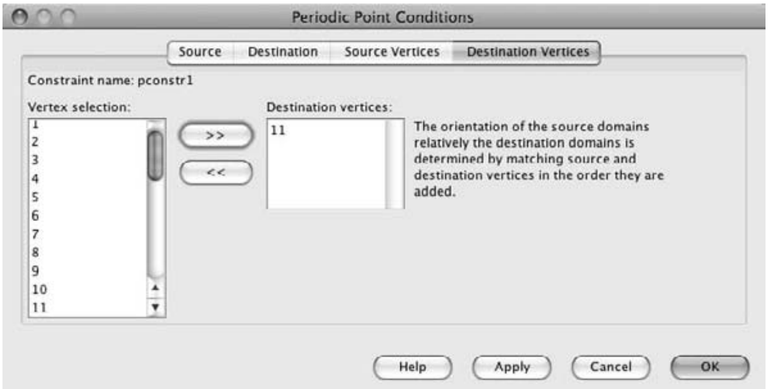


FIGURE 7.136 2D_ACGS_1 model Periodic Point Conditions, Destination Vertices (11) page

Click the Destination Vertices tab. Click the >> button to select point 11 as a destination vertex. See Figure 7.136. Click OK.

Physics Settings: Global Equations

Using the menu bar, select Physics > Global Equations. In the Global Equations edit windows, enter the information shown in Table 7.25. Click OK. See Figure 7.137.

Table 7.25 Global Equations Edit Window

Name	Equation f(u,ut,utt,t)	Init (u)	Init (ut)	Description	Figure Number
omega	omegatt-(M+Tz)/(I+I0)	0	0	Rotation angle	7.137

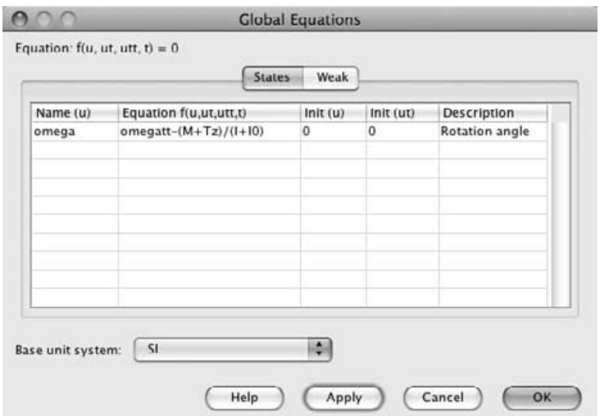


FIGURE 7.137 2D_ACGS_1 model Global Equations edit window

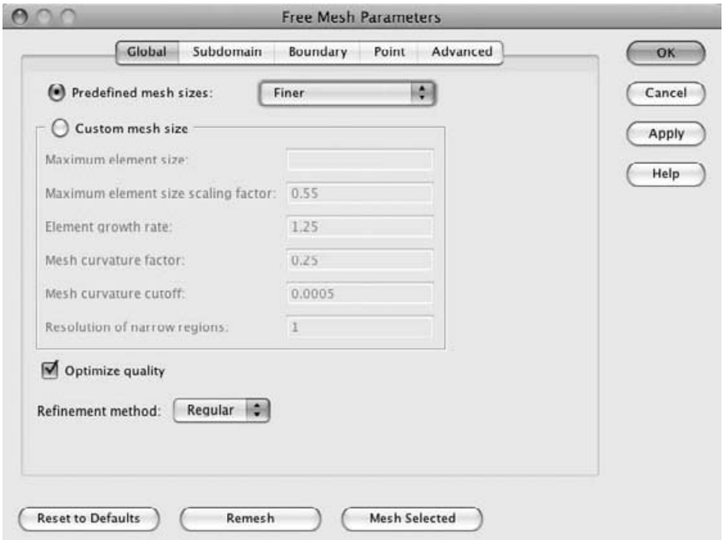


FIGURE 7.138 2D_ACGS_1 model Free Mesh Parameters edit window

Mesh Generation

From the toolbar, select Mesh > Free Mesh Parameters > Global. Select Predefined mesh sizes > Finer (from the pull-down list). See Figure 7.138.

Click the Boundary tab. Select boundaries 2, 5–8, 10–12, 14, 19–21. Enter 0.005 in the Maximum element size window. See Figure 7.139.

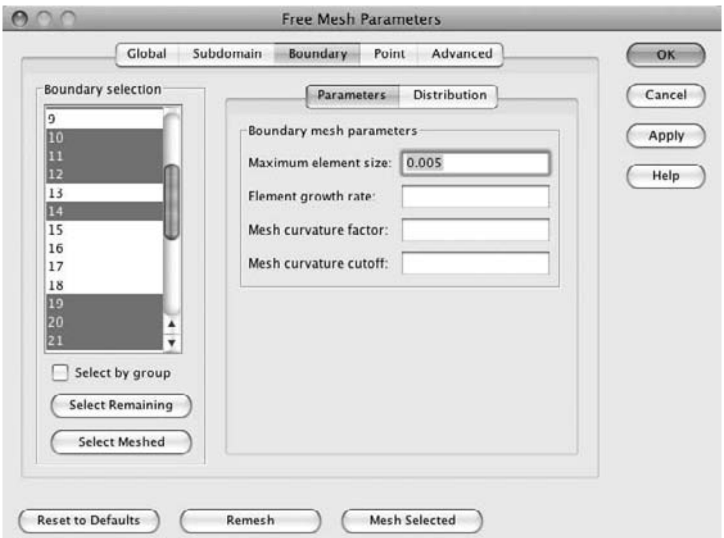


FIGURE 7.139 2D_ACGS_1 model Free Mesh Parameters, Boundary page

FIGURE 7.140 2D_ACGS_1 model Copy Mesh button

Click the Mesh Selected button.

Select boundaries 1 and 2. Click the Copy Mesh button on the Mesh toolbar. See Figure 7.140.

NOTE The purpose of copying the mesh from one edge to the other edge is to ensure that the proper phase relationship is maintained across the mesh. Otherwise, there might be a mismatch between the edges, which would distort the mesh and possibly result in solution problems.

Select boundaries 3 and 7. Click the Copy Mesh button.

Select boundaries 4 and 8. Click the Copy Mesh button.

Select boundaries 15 and 19. Click the Copy Mesh button.

Select boundaries 16 and 20. Click the Copy Mesh button, and then click OK. See Figure 7.141.

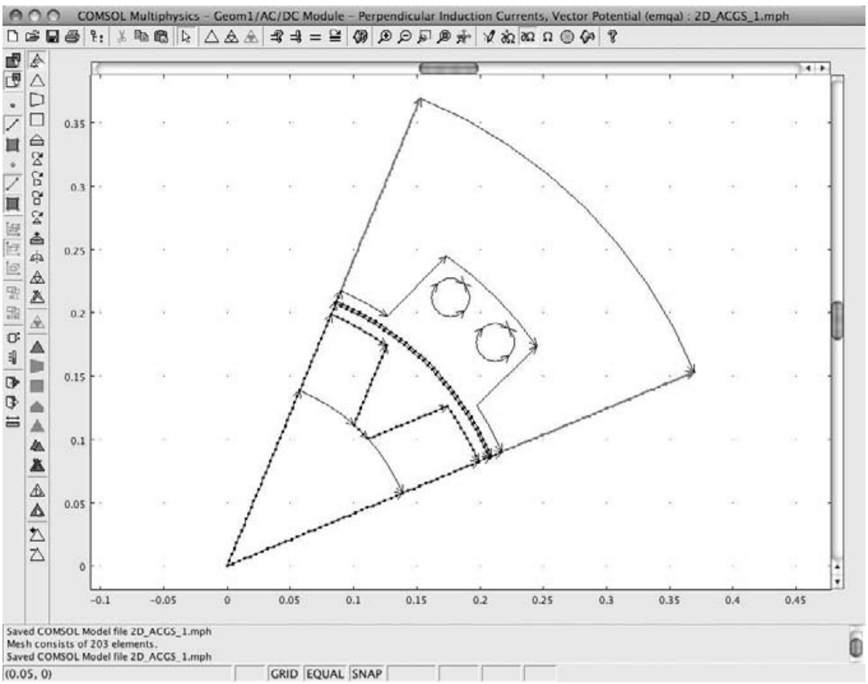


FIGURE 7.141 2D_ACGS_1 model copied meshed boundaries

FIGURE 7.142 2D_ACGS_1 model Mesh Remaining (Free) button

Click the Mesh Remaining (Free) button on the Mesh toolbar. See Figure 7.142. Figure 7.143 shows the meshed model.

Solving the Static 2D_ACGS_1 Model

Using the menu bar, Select Solve > Solver Parameters.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

FIGURE 7.143 2D_ACGS_1 model meshed

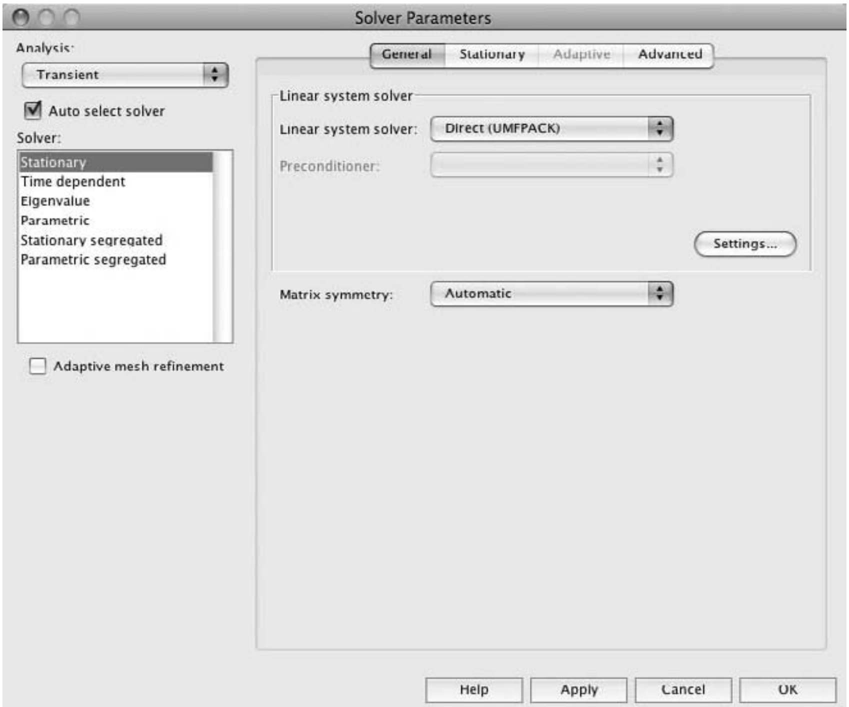


FIGURE 7.144 2D_ACGS_1 model Solver Parameters edit window

Select Stationary in the Solver selection window. See Figure 7.144. Click OK.

Using the menu bar, select Solve > Solver Manager > Solve For. Unselect the variable omega under ODE(OD). See Figure 7.145. Click OK.

Select Solve > Solve Problem. Click the Save button.

Select File > Save As. Enter 2D_ACGS_2.mph in the Save As edit window. See Figure 7.146.

Click the Save button. See Figure 7.147.

NOTE The 2D_ACGS_1 model (static) solution was built to gain experience in the creation of a complex geometrical sector model. It was saved as 2D_ACGS_2.mph and will act as the initial estimate for the 2D_ACGS_2 model (transient) solution.

2D AC Generators, Static Sector Model: Summary and Conclusions

The static 2D AC generator sector models has now been built. This model allows the modeler to solve the AC generator model through the use of a simpler geometric representation. It avoids some of the potential problems that might be observed at run

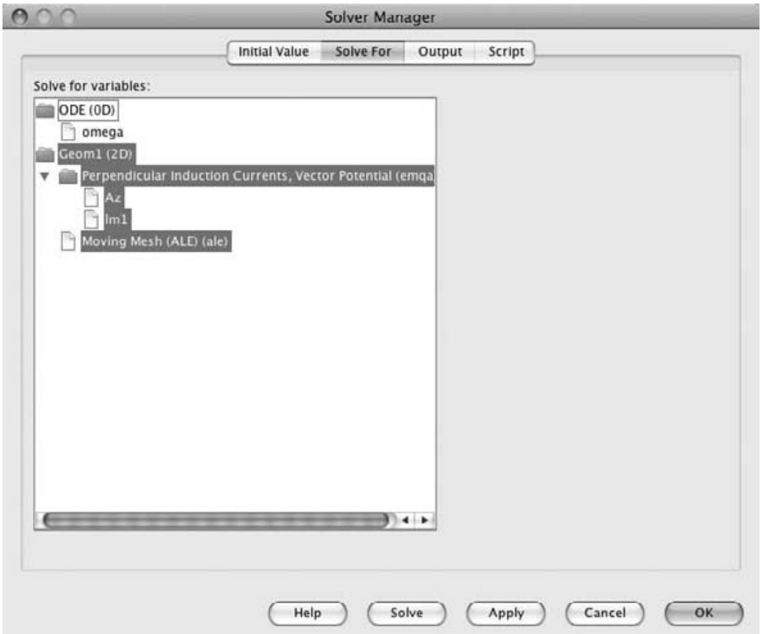


FIGURE 7.145 2D_ACGS_1 model with omega unselected

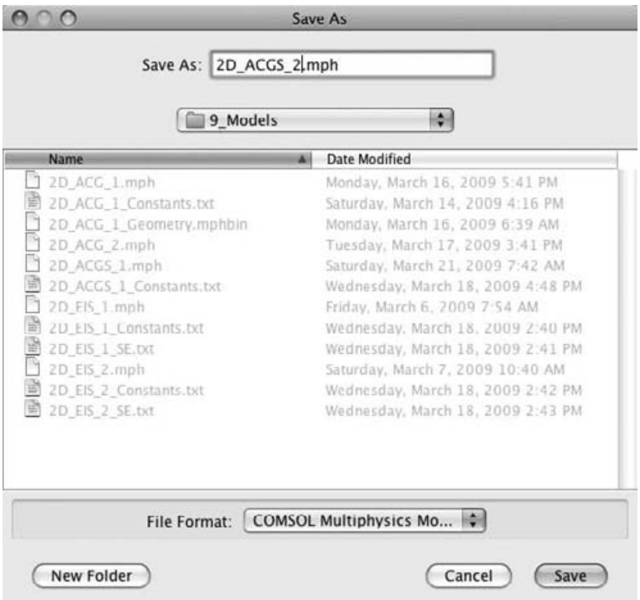


FIGURE 7.146 2D_ACGS_2 model Save As edit window

FIGURE 7.147 2D_ACGS_2 model initial solution

time. This model generates low-frequency (60 Hz) current and voltage, as would be typically found on the power transmission grid. This model demonstrates the use of both hard (not easily magnetized) and soft (easily magnetized) nonlinear magnetic materials in the construction of a rotating machine for the conversion of mechanical energy to electrical energy.

2D AC Generator Sector Model (2D_ACGS_2): Transient

The numerical solution model presented in this subsection (2D_ACGS_2) follows directly from the earlier 2D_ACGS_1 model, which we just built. In this subsection, the transient 2D generator sector model (2D_ACGS_2) utilizes the static 2D_ACGS_1 model as the initial solution to the transient problem. The new version avoids all the complex geometrical building by starting with the earlier saved solution.

To start building the 2D_ACGS_2 model (transient) solution, activate the COMSOL Multiphysics software. In the Model Navigator, click the Open tab. Select "2D_ACGS_2.mph." See Figure 7.148.

Click OK. See Figure 7.149.

FIGURE 7.148 2D_ACGS_2 Model Navigator, initial solution selection

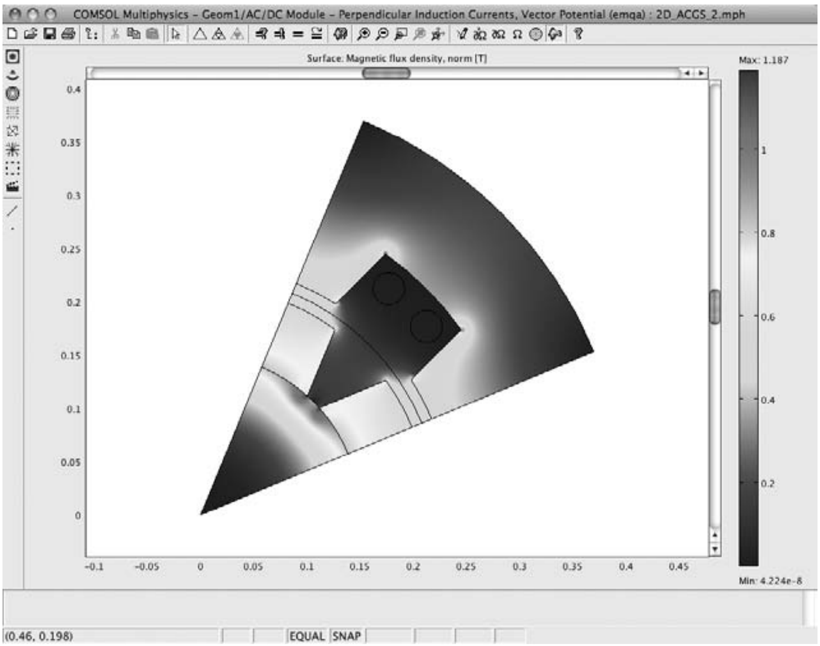


FIGURE 7.149 2D_ACGS_2 Model Navigator, initial solution

NOTE Because the initial solution to the 2D_ACG_2 model (transient) has already been built and verified, the modeler can proceed directly to implementing the necessary transient solver setup parameters.

Solving the Transient 2D_ACGS_2 Model

Using the menu bar, select Solve > Solver Parameters. select "Transient" from the Analysis pull-down list.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Select "Time dependent" in the Solver selection window. In the Times edit window, enter linspace(0,2.6,131). For later versions of COMSOL Multiphysics software enter range(0,2.6/130,2.6). In the Absolute tolerance edit window, enter Az 1e-3 lm1 5e3 omega 0.015. Click the Apply button. See Figure 7.150.

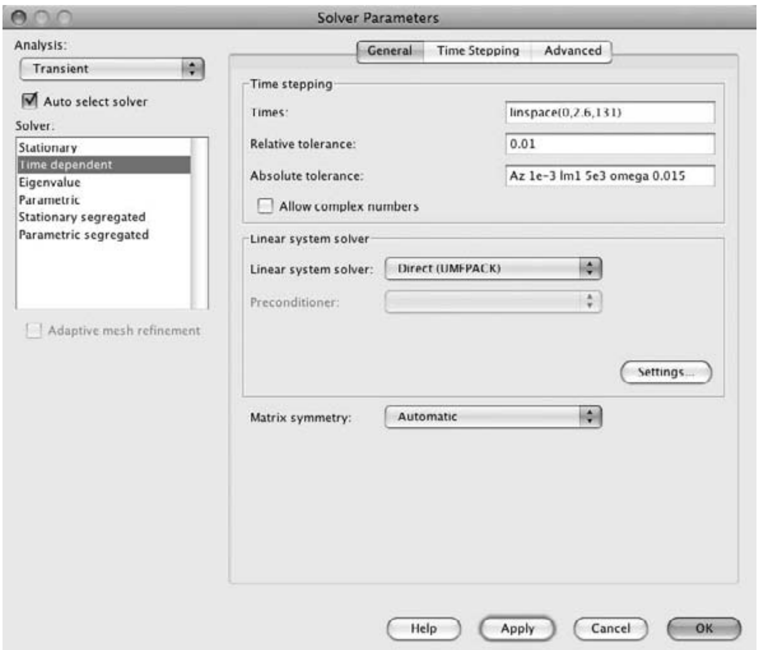


FIGURE 7.150 2D_ACGS_2 model Solver Parameters edit window

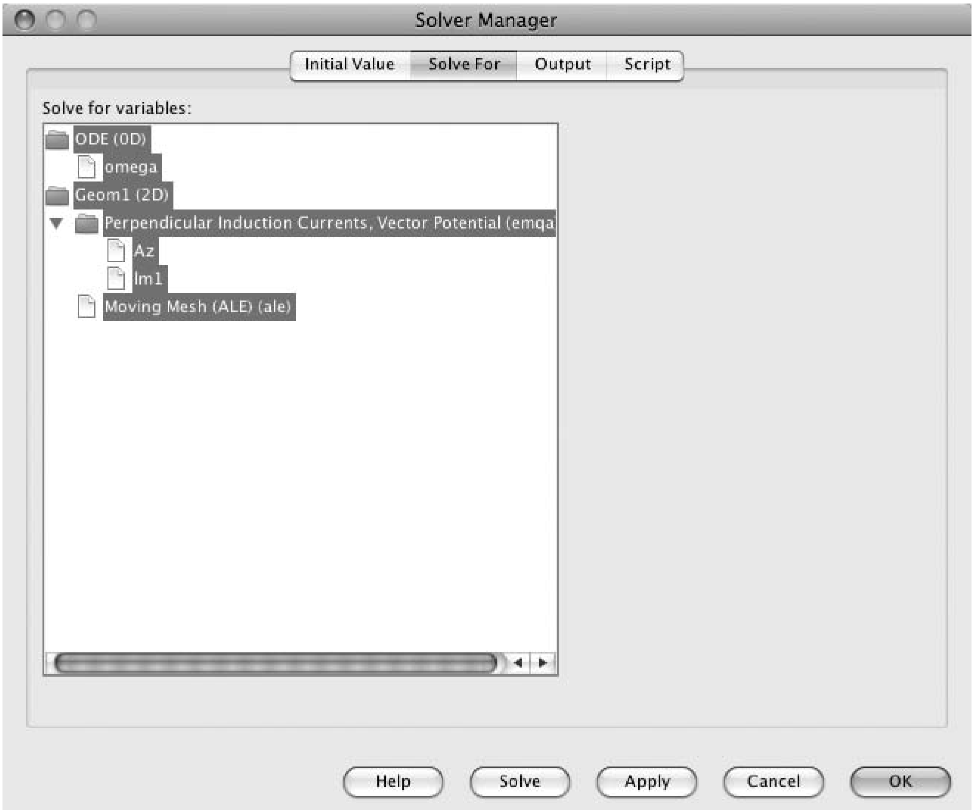


FIGURE 7.151 2D_ACGS_2 model Solver Manager, Solve For page

NOTE It is important to enter the solver parameters *exactly* as specified: otherwise, the modeler will see error messages.

Click OK. Using the menu bar, select Solve > Solver Manager > Solve For. See Figure 7.151.

Select all variables. Click OK.

Using the menu bar, select Solve > Restart. See Figure 7.152.

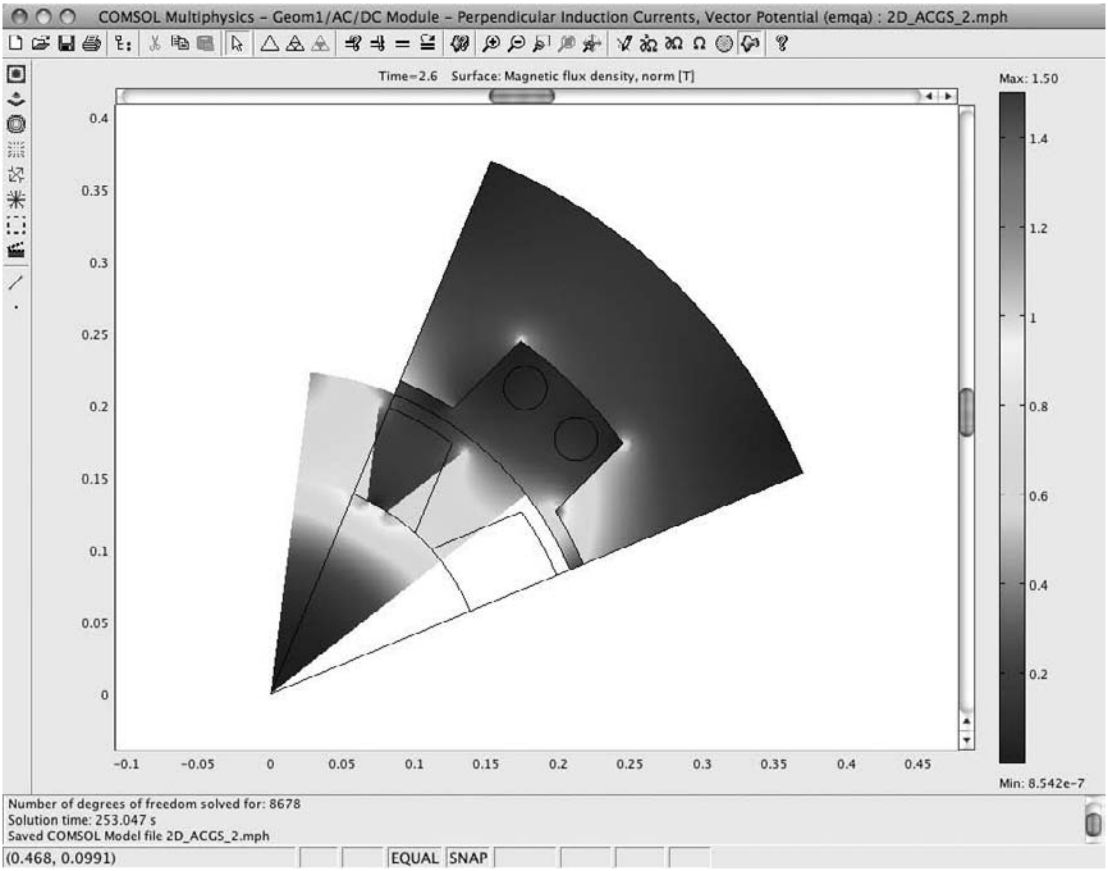


FIGURE 7.152 2D_ACGS_2 model solution default plot, final frame

Postprocessing and Visualization

Select Postprocessing > Plot Parameters > General. Check the Surface and Contour check boxes under Plot type. Select "Solution at time 0.2 seconds" from the Solutions to use pull-down list. See Figure 7.153.

FIGURE 7.153 2D_ACGS_2 model plot Parameters, General tab

Click the Surface tab. Select “Magnetic flux density, norm.” See Figure 7.154.

Click the Contour tab. Enter Az in the Expression edit window. Enter 15 in the Levels edit window. Click the Uniform color radio button. Click the Color button, and select black. See Figure 7.155.

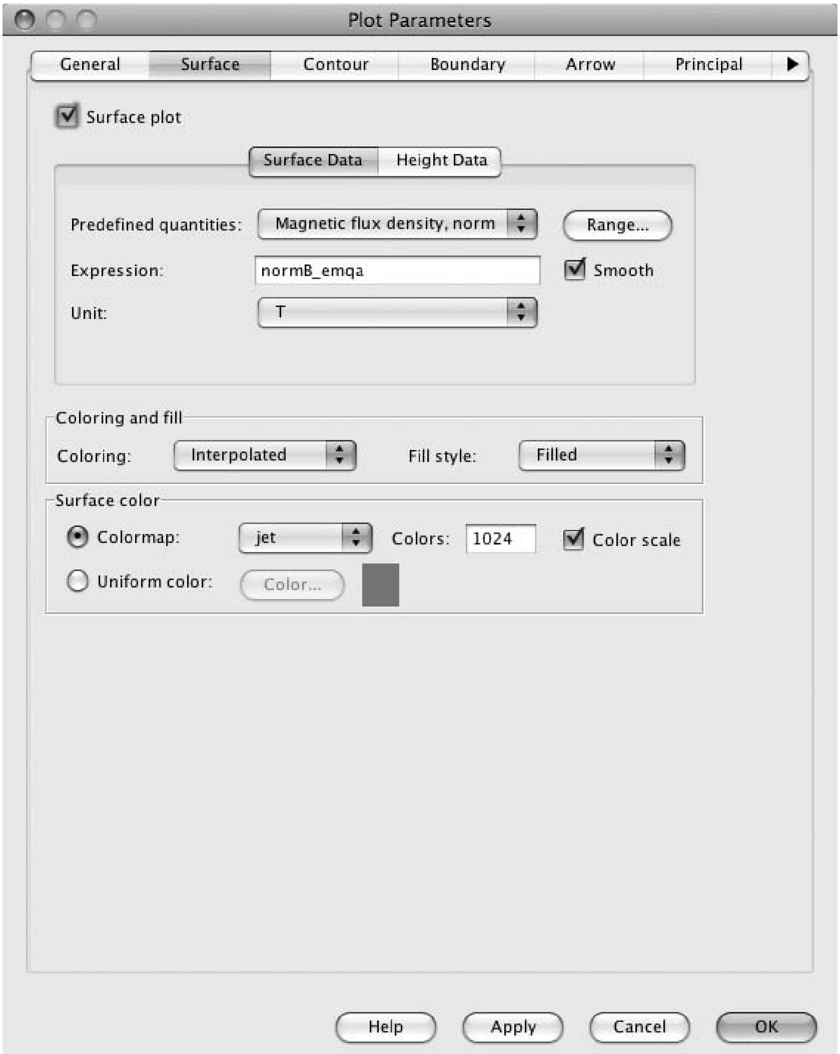


FIGURE 7.154 2D_ACGS_2 model Plot Parameters, Surface tab

FIGURE 7.155 2D_ACGS_2 model Plot Parameters, Contour tab

Click OK. See Figure 7.156.

To view the rotation angle as a function of time, select Postprocessing > Global Variables Plot. Select “Rotation angle” from the Predefined quantities list. Click the > button. See Figure 7.157.

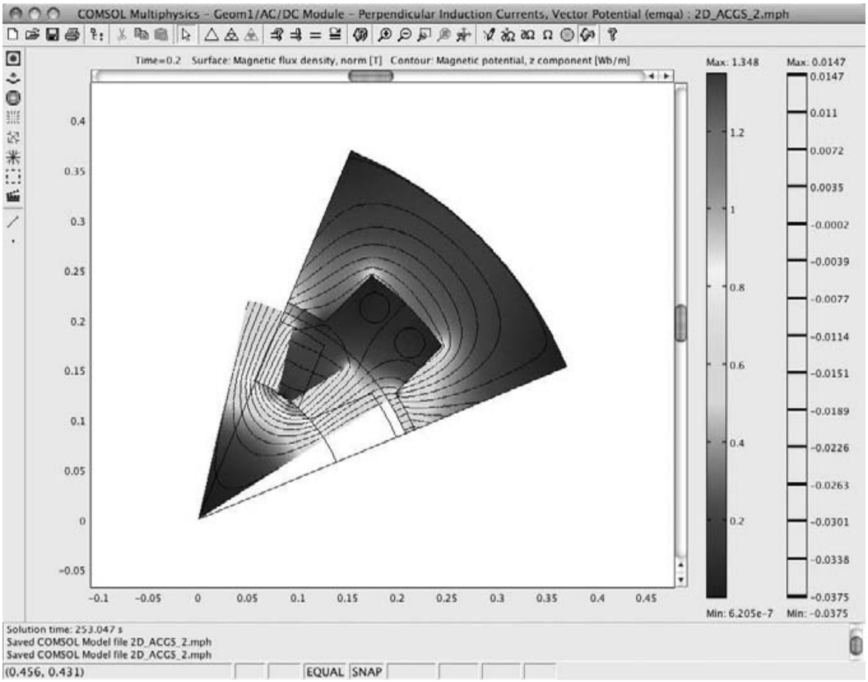


FIGURE 7.156 2D_ACGS_2 model magnetic flux density and magnetic potential, *z* component

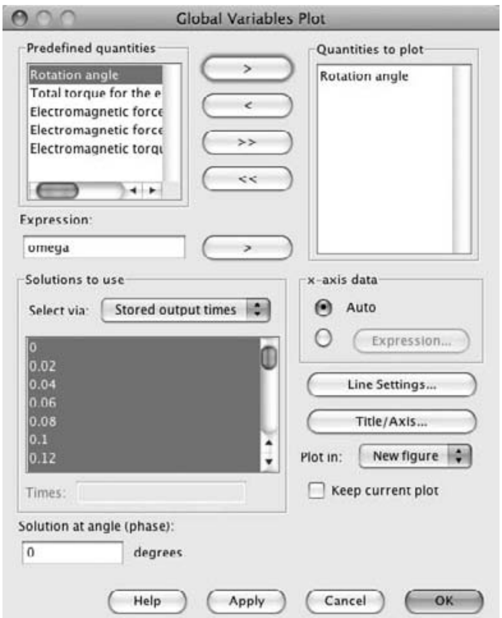


FIGURE 7.157 2D_ACGS_2 model Global Variables Plot edit window

FIGURE 7.158 2D_ACGS_2 model global variables plot

Click the Apply button. See Figure 7.158. Click OK.

Postprocessing Animation

Select Postprocessing > Plot Parameters > Animate. Select the solutions from 0.0 to 0.7 in the Solutions to use window. See Figure 7.159.

Click the Start Animation button. See Figure 7.160.

Alternatively, you can play the file Movie7_ACGS_2.avi that was supplied with this book.

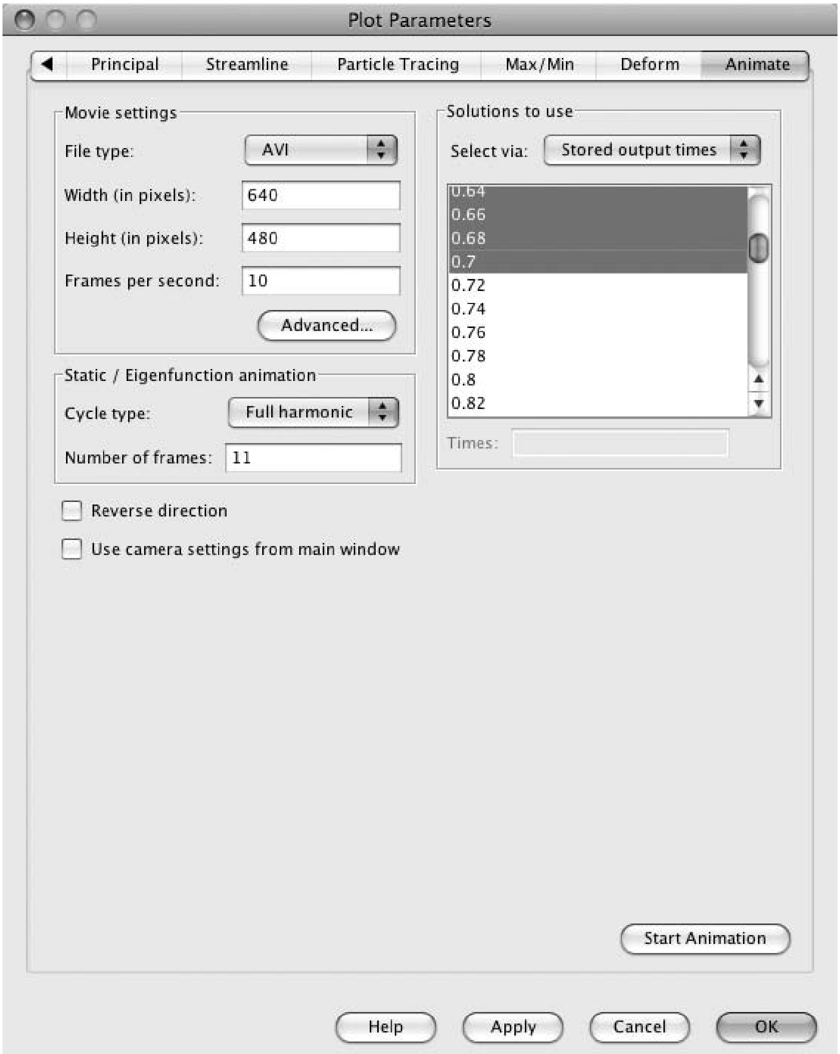


FIGURE 7.159 2D_ACGS_2 model Plot Parameters, Animate tab

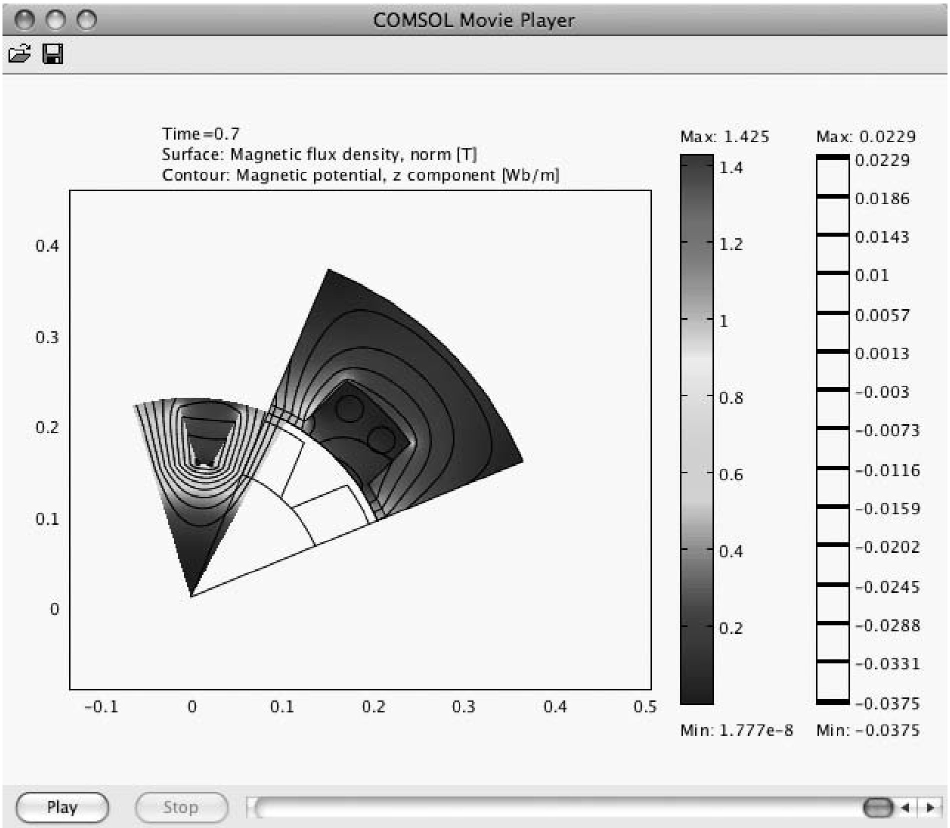


FIGURE 7.160 2D_ACGS_2 model animation, final frame

2D AC Generators, Static and Transient Models: Summary and Conclusions

The two 2D AC generator sector models—static and transient—have now been built. These models generate low-frequency (60 Hz) current and voltage, as would be typically found on the power transmission grid. These models demonstrate the use of both hard (not easily magnetized) and soft (easily magnetized) nonlinear magnetic materials in the construction of a rotating machine for the conversion of mechanical energy to electrical energy.

The modeling and physics concepts employed in this section of Chapter 7 include addition of an ordinary differential equation (ODE), mechanical to electrical energy conversion, hard and soft nonlinear magnetic materials, mesh mapping, moving mesh (ale), and geometric assembly (pair creation across a boundary).

References

1. http://en.wikipedia.org/wiki/Electrical_resistivity_tomography
2. http://en.wikipedia.org/wiki/Industrial_process_imaging
3. http://en.wikipedia.org/wiki/Electrical_impedance_tomography
4. http://en.wikipedia.org/wiki/Alternating_current
5. P. Manwaring et al., Arbitrary Geometry Patient Interfaces for Breast Cancer Detection and Monitoring with Electrical Impedance Tomography, *Proceedings of the 30th Annual International IEEE EMBS Conference,* 2008, pp. 1178–1180.
6. X. Chen et al., Lung Ventilation Monitoring Incorporating Prior Information by Electrical Impedance Tomography, *Proceedings of I2MTC 2008: IEEE International Instrumentation and Measurement Technology Conference,* 2008, pp. 1531–1536.
7. D. S. Holder, Electrical Impedance Tomography of Brain Function, *Proceedings of Automation Congress 2008,* pp. 1–6.
8. http://en.wikipedia.org/wiki/Generators
9. http://en.wikipedia.org/wiki/Ordinary_differential_equation
10. http://en.wikipedia.org/wiki/Electrical_impedance
11. http://en.wikipedia.org/wiki/Ohm%27s_law
12. http://en.wikipedia.org/wiki/Oliver_Heaviside
13. http://en.wikipedia.org/wiki/Arthur_Kennelly
14. W. T. Scott, *The Physics of Electricity and Magnetism* (2nd ed.), John Wiley and Sons, 1966, Chapter 9.2–9.4, pp. 461–485.
15. http://en.wikipedia.org/wiki/Angular_frequency
16. http://en.wikipedia.org/wiki/Reactance_(electronics)
17. http://en.wikipedia.org/wiki/Skin_depth
18. http://en.wikipedia.org/wiki/Permittivity
19. http://en.wikipedia.org/wiki/Michael_Faraday
20. http://en.wikipedia.org/wiki/Electromagnetic_induction
21. http://en.wikipedia.org/wiki/Thomas_Alva_Edison
22. http://en.wikipedia.org/wiki/George_Westinghouse
23. http://en.wikipedia.org/wiki/Nikola_Tesla
24. http://en.wikipedia.org/wiki/War_of_the_currents
25. http://en.wikipedia.org/wiki/Joule%27s_Law

26. http://en.wikipedia.org/wiki/Transformer

27. *COMSOL Multiphysics Modeling Guide*, Using Assemblies, Version 3.4, October 2007, COMSOL AB, Stockholm, Sweden, pp. 351–367.

28. http://en.wikipedia.org/wiki/Ordinary_differential_equation

Exercises

1. Build, mesh, and solve the basic 2D electric impedance sensor model problem presented in this chapter.
2. Build, mesh, and solve the advanced 2D electric impedance sensor model problem presented in this chapter.
3. Build, mesh, and solve the static 2D AC generator model (2D_ACG_1) problem presented in this chapter.
4. Build, mesh, and solve the transient 2D AC generator model (2D_ACG_2) presented in this chapter.
5. Build, mesh, and solve the static 2D AC generator sector model (2D_ACGS_1) presented in this chapter.
6. Build, mesh, and solve the transient 2D AC generator sector model (2D_ACGS_2) presented in this chapter.
7. Explore other materials as applied in the 2D electric impedance sensor models.
8. Explore other materials as applied in the 2D AC generator models.
9. Explore adding more turns to the 2D AC generator models.
10. Explore how the 2D electric impedance sensor model might be used to discover voids in boats, airplanes, bridges, and other areas.

8 3D Modeling

In This Chapter

■ 3D Modeling Guidelines for New COMSOL® Multiphysics® Modelers

3D Modeling Considerations

In this chapter on 3D modeling, all the basic material on 2D modeling presented in the earlier chapters will be assumed, utilized, and expanded. In the earlier chapters, models were built and solved using static, quasi-static, and transient methods. In this chapter,

the methods employed will be either static or quasi-static. The level of model solution complexity (difficulty) is increased through the development of models that explore applied physics at a more realistic and difficult level. In the three 3D models developed in this chapter, three modeling concept areas are explored: large dimensional differences, electrostatic field mapping, and magnetostatic field mapping. Each of these areas has broad industrial and scientific modeling applicability and potential levels of complexity.

The 3D models in this chapter implicitly assume, in compliance with the laws of physics, that the energy flow, the materials properties, the environment, and any other conditions and variables of interest are homogeneous, isotropic, or constant, unless otherwise specified (e.g., time dependent), throughout the entire domain of interest, both within the model and, through the boundary conditions, in the environs of the model.

Three models and variations are presented here: the 3D thin layer resistance models, the 3D electrostatic potential between cylinders models, and the 3D magnetic field of a helmholtz coil models. The first two models are developed using application modes from the basic COMSOL® Multiphysics® software. The Helmholtz coil model requires the AC/DC Module. Each of these three models introduces the modeler to different modeling aspects in the employment of the basic COMSOL Multiphysics software and the AC/DC Module to explore a range of diverse design, test, and engineering problems.

The 3D thin layer resistance models explore the modeling of a technology that is widely employed in both research and applied development for science and industry. Both thin[1,2] and thick[3] layers (coatings) are widely applied. Layers, such as those modeled here or other layers that may inadvertently or unknowingly form, can significantly modify the overall performance (behavior) of a device structure.

NOTE When building a model, the modeler should perform at least a first estimate review of the conditions to which the modeled structure will be exposed under the normal (extreme) conditions of use. In that review, a variety of questions should be asked, including these possibilities: Will corrosion films form? Will any of the structure melt? Will any of the films exhibit a structural transition? Will any of the films exhibit an electronic/magnetic properties shift?

3D Coordinate System

In a steady-state solution to a 3D model, parameters can vary only as a function of position in space (x), space (y), and space (z) coordinates. Such a 3D model represents the parametric condition of the model in a time-independent mode (quasi-static).

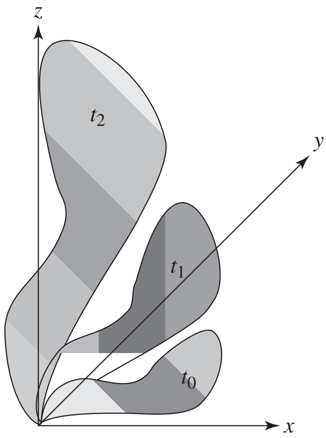


FIGURE 8.1 3D coordinate system, plus time

In a transient solution model, parameters can vary both by position in space (x), space (y), space (z) and in time (t). See Figure 8.1.

The transient solution model is essentially a sequential collection of (quasi-static) solutions, except that one or more of the dependent variables [$f(x, y, z, t)$] has changed with time. The space coordinates (x), (y), and (z) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values ($x_{min} <= x <= x_{max}$), ($y_{min} <= y <= y_{max}$), and ($z_{min} <= z <= z_{max}$) The time coordinate (t) represents the range of values ($t_{min} <= t <= t_{max}$) from the beginning of the observation period (t_{min}) to the end of the observation period (t_{max}).

Electrical Resistance Theory

A well-known example of the application of thin layer technology is the touch screen,[4] which is widely used in computers, personal digital assistants (PDAs), electronic lock pads, and other devices. The fundamental concept of the touch screen is relatively simple. The underlying touch screen principle starts with Ohm's law.

NOTE Ohm's law was discovered by Georg Ohm and published in 1827:

$$I = \frac{V}{R} \tag{8.1}$$

where I = current in amperes (A)
V = voltage (electromotive force) in volts (V)
R = resistance in ohms

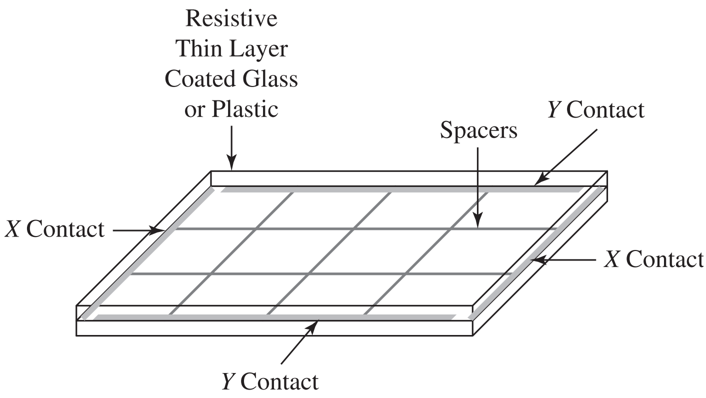


FIGURE 8.2 3D touch screen geometry

This technology utilizes the concept of an electrical resistance[5] divider[6] (voltage divider) to locate the point of contact (touch). When implemented in four-wire touch screen technology, this technology employs two thin layer orthogonal voltage dividers. In the touch screen technology, the thin layer resistive sheets are coated onto an insulating glass or plastic substrate. The substrates are mounted with the thin resistive layers facing each other and separated by an array of thin insulating dots, insulating bars, or a similar porous insulating spacer array. See Figure 8.2.

When pressure (touch) is applied to the screen, a point of contact forms at that location. See Figure 8.3.

The voltage V_{XT} is measured as shown in Figure 8.3 (measurement circuitry not shown). V_{YT} is similarly measured sequentially. The X location of the contact point is

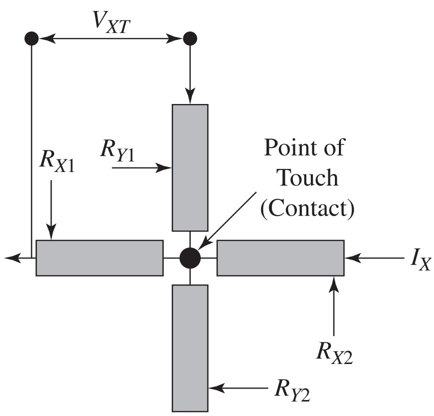


FIGURE 8.3 3D touch screen divider circuit

determined as follows:

$$V_{XT} = I_X * R_{X1} : V_{\mathrm{TOTAL}} = I_X * (R_{X1} + R_{X2}) \tag{8.2}$$

where I_X = X current in amperes (A)
V_{XT} = voltage drop (electromotive force) in volts (V)
V_{TOTAL} = total voltage (electromotive force) along X in volts (V)
R_{X1} = divider resistance 1 in ohms
R_{X2} = divider resistance 2 in ohms

since

$$R = \frac{\rho L}{A} \tag{8.3}$$

where R = resistance in ohms
ρ = resistivity in ohm*m
L = length of the resistive material (m)
A = cross-sectional area in meters squared (m^2)

Thus the length (distance) to the contact point is

$$\frac{V_{XT}}{V_{\mathrm{TOTAL}}} * (L) = \frac{R_{X1}}{R_{X1} + R_{X2}} * (L) = \frac{L_{X1}}{L_{X1} + L_{X2}} * (L) \tag{8.4}$$

where V_{XT} = voltage drop (electromotive force) in volts (V)
V_{TOTAL} = total voltage (electromotive force) along X in volts (V)
R_{X1} = divider resistance 1 in ohms
R_{X2} = divider resistance 2 in ohms
L_{X1} = resistor length 1 in meters (m)
L_{X2} = resistor length 2 in meters (m)
$L = L_{X1} + L_{X2}$ in meters (m)

Thin Layer Resistance Modeling Basics

The first example presented here, the 3D thin layer resistance model, thin layer approximation (3D_TLR_1 model), explores the use of the thin layer approximation in the solution of a direct current conduction model. In the problem explored in both this model and the model to follow, the current balance throughout the domains is described as follows:

$$V \cdot (-\sigma \nabla V) = 0 \tag{8.5}$$

where V = electric potential (electromotive force) in volts (V)
Δ = gradient
σ = electrical conductivity in siemens per meter (S/m)

In the thin layer approximation model, it is assumed that the x and y components of the current density vector are sufficiently small in the thin layer that only the z component makes a contribution. Thus

$$-\sigma * \frac{d^2V}{dz^2} = 0 \tag{8.6}$$

where V = electric potential (electromotive force) in volts (V)
σ = electrical conductivity in siemens per meter (S/m)

By substitution, it can be seen that

$$V = \alpha * z + \beta \tag{8.7}$$

where α = constant (V/m)
β = constant (V)

Equation 8.7 is one possible solution for Equation 8.6.

NOTE Considering the following:

$$\frac{dV}{dz} = \alpha : \frac{d^2V}{dz^2} = 0 \tag{8.8}$$

Assuming that

$$V_{\text{lower}} = V_1 : V_{\text{upper}} = V_2 \tag{8.9}$$

then for $z = 0$:

$$\beta = V_1 \tag{8.10}$$

For $z = \delta$:

$$\alpha = \frac{V_2 - V_1}{\delta} \tag{8.11}$$

where δ = thickness of the thin layer in meters (m).

Because

$$J_Z = -\sigma * \frac{dV}{dz} = -\sigma * \alpha = -\sigma * \left(\frac{V_2 - V_1}{\delta}\right) \tag{8.12}$$

and there are no sources or sinks, J_z is the current flow through the system.

NOTE The use of the thin layer approximation is applicable to any problem in which flow is described by the divergence of a gradient flux (i.e., diffusion, heat conduction, flow through porous media under Darcy's law).

The application of the thin layer approximation is especially valuable to the modeler when the differences in domain thickness are so great that the mesh generator is unable to properly mesh the model or creates more elements than the modeling platform can handle (the "run out of memory" problem). In those cases, this approximation may enable a model to be solved that would otherwise fail.

3D Thin Layer Resistance Model: Thin Layer Approximation

The following numerical solution model (3D_TLR_1 model) is derived from a model that was originally developed by COMSOL as a Multiphysics Electromagnetics demonstration model. That model was developed for distribution with the Multiphysics software as a COMSOL Multiphysics Model Library.

To start building the 3D_TLR_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select COMSOL Multiphysics > Electromagnetics > Conductive Media DC. See Figure 8.4. Click OK.

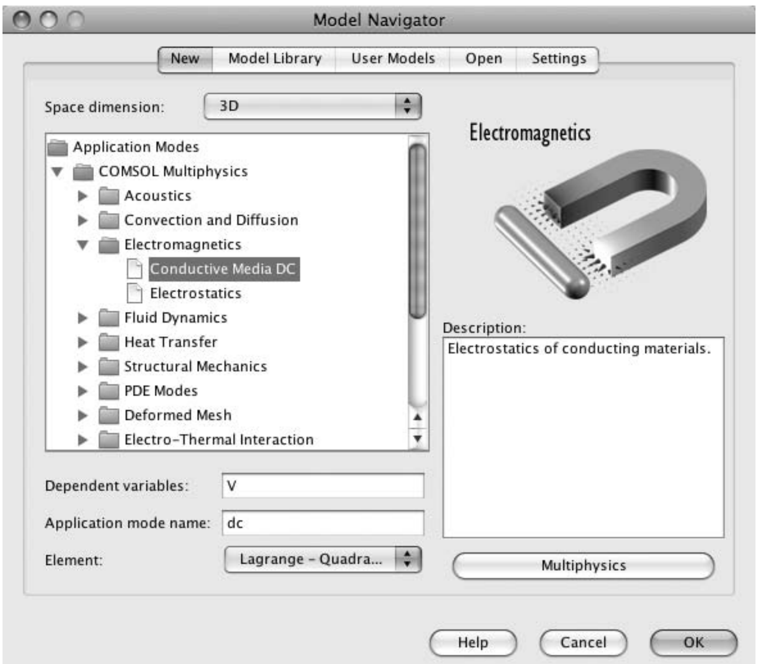


FIGURE 8.4 3D_TLR_1 Model Navigator setup

Table 8.1 Geometry Components

Name	Style	Base	Length (X, Y, Z)	Axis Base Point (X, Y, Z)	Figure Number
BLK1	Solid	Corner	(1, 1, 0.1)	(0, 0, 0)	8.5
BLK2	Solid	Corner	(1, 1, 0.1)	(0, 0, 0.1)	8.6

Geometry Modeling

Using the menu bar, select Draw > Block. In the Block edit window, enter the information shown in Table 8.1. Click OK after filling in the parameters of each separate block in the Block edit window. See Figures 8.5 and 8.6.

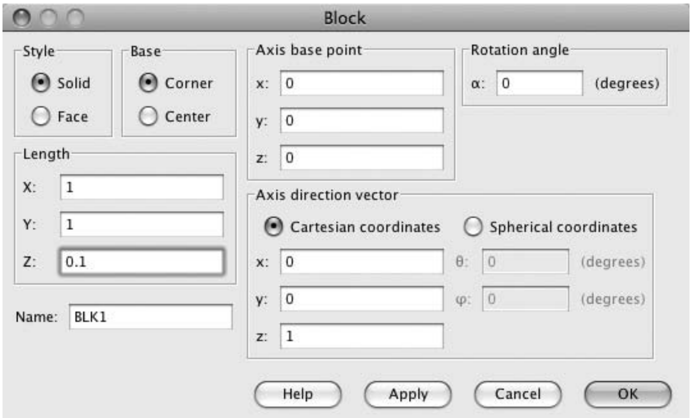


FIGURE 8.5 3D_TLR_1 model BLK1 edit window

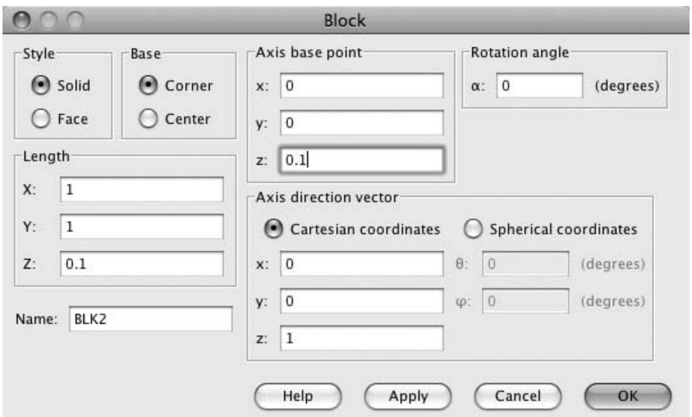


FIGURE 8.6 3D_TLR_1 model BLK2 edit window

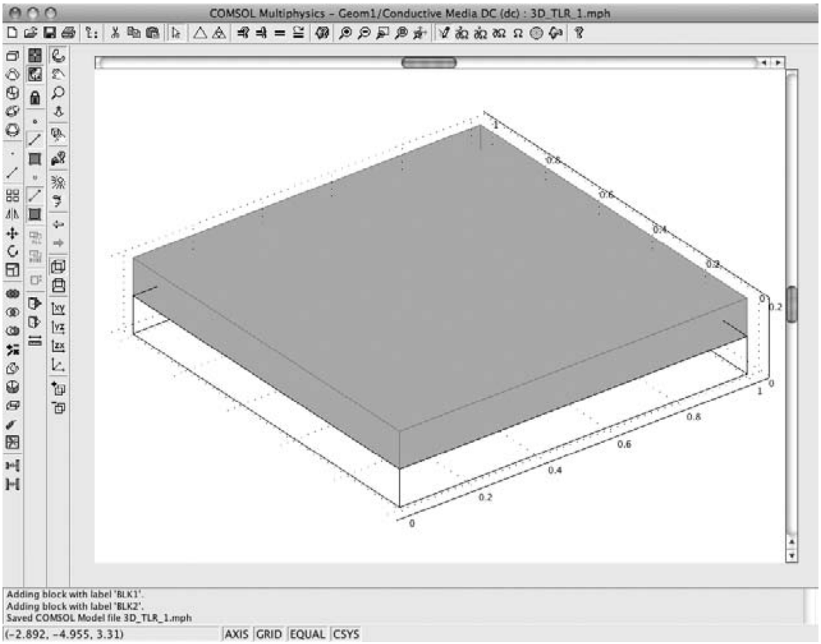


FIGURE 8.7 3D_TLR_1 model BLK2

Click the Zoom Extents button. See Figure 8.7.

Select File > Save As. Enter 3D_TLR_1 in the Save As edit window. See Figure 8.8. Click the Save button.

Using the menu bar, select Draw > Work-Plane Settings. See Figure 8.9.

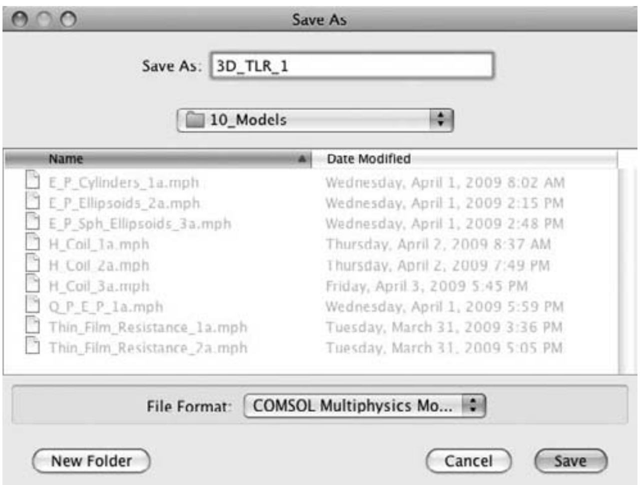


FIGURE 8.8 3D_TLR_1 model Save As edit window

FIGURE 8.9 3D_TLR_1 model Work-Plane Settings edit window

NOTE The use of the Work-Plane Settings is intended to make specific 2D planes available to the modeler to facilitate the creation of essentially 2D objects in the 3D geometry.

Click OK, using the default settings. Click the Zoom Extents button. See Figure 8.10.

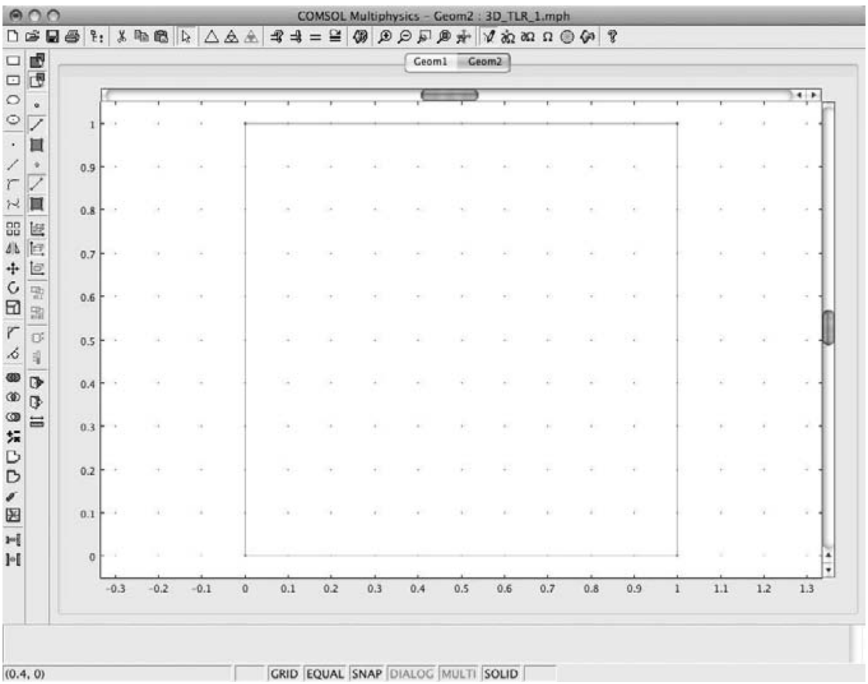


FIGURE 8.10 3D_TLR_1 model Geom2 work-plane

FIGURE 8.11 3D_TLR_1 model Circle edit window

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.6, set the Base as Center, and set x equal to 0 and y equal to 1. See Figure 8.11. Click OK.

Using the menu bar, select Draw > Specify Objects > Square. Enter a width of 1, set the Base as Corner, and set x equal to 0 and y equal to 0. See Figure 8.12. Click OK.

Using the menu bar, select Draw > Create Composite Object.

Enter C1* SQ1 in the Set formula edit window. See Figure 8.13.

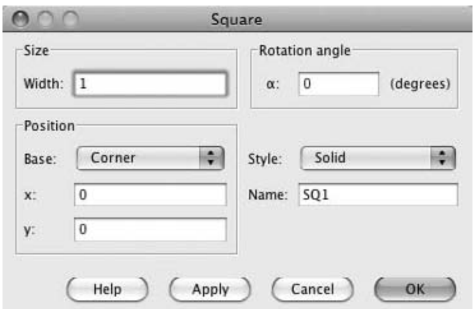


FIGURE 8.12 3D_TLR_1 model Square edit window

FIGURE 8.13 3D_TLR_1 model Create Composite Object edit window

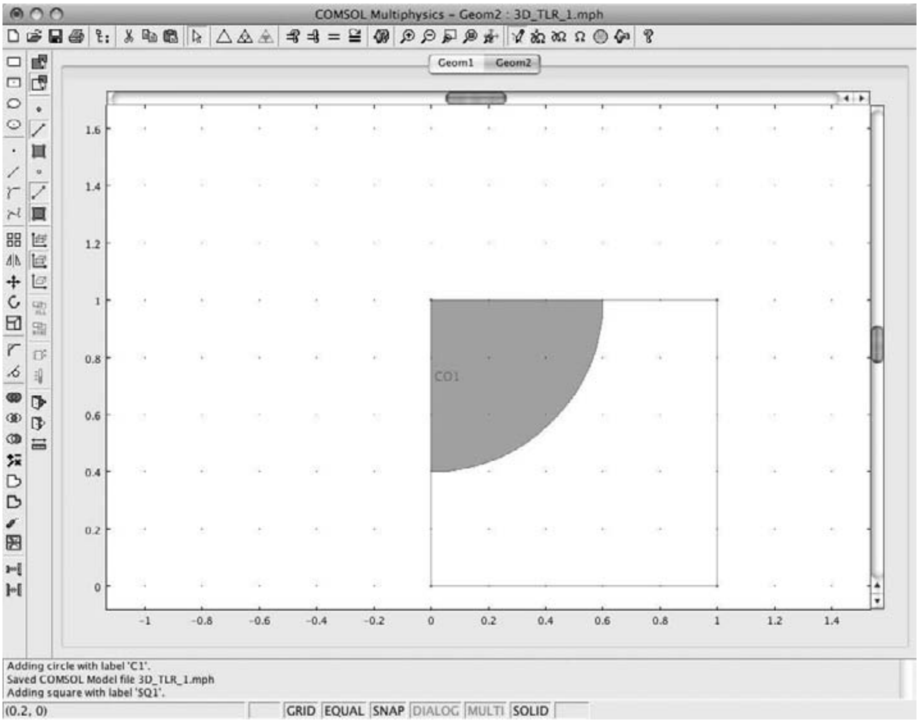


FIGURE 8.14 3D_TLR_1 model CO1 (intersection of circle and square)

NOTE The formula *X* * *Y* creates the intersection of *X* and *Y*.

Click OK. See Figure 8.14.
Using the menu bar, select Draw > Embed. See Figure 8.15. Click OK.

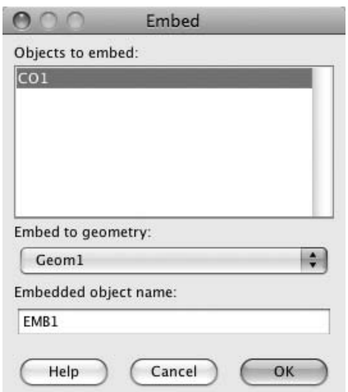


FIGURE 8.15 3D_TLR_1 model Embed edit window

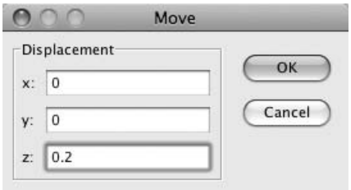


FIGURE 8.16 3D_TLR_1 model Move edit window

As is obvious, the quarter-circle electrode needs to be moved to the upper surface of the upper block. Using the menu bar, select Draw > Modify > Move. Enter x = 0, y = 0, and z = 0.2. See Figure 8.16.

Click OK. See Figure 8.17.

Select EMB1 and BLK2 (click on EMB1 and then Shift-click on BLK2). See Figure 8.18. Using the menu bar, select Draw > Coerce To > Solid.

Using the menu bar, select Draw > Create Pairs. Select BLK1 and CO1. See Figure 8.19.

Click OK. See Figure 8.20.

Physics Subdomain Settings: Conductive Media DC (dc)

Having established the geometry for the 3D_TLR_1 model of two blocks, an electrode, and an identity paired interface, the next step is to define the fundamental

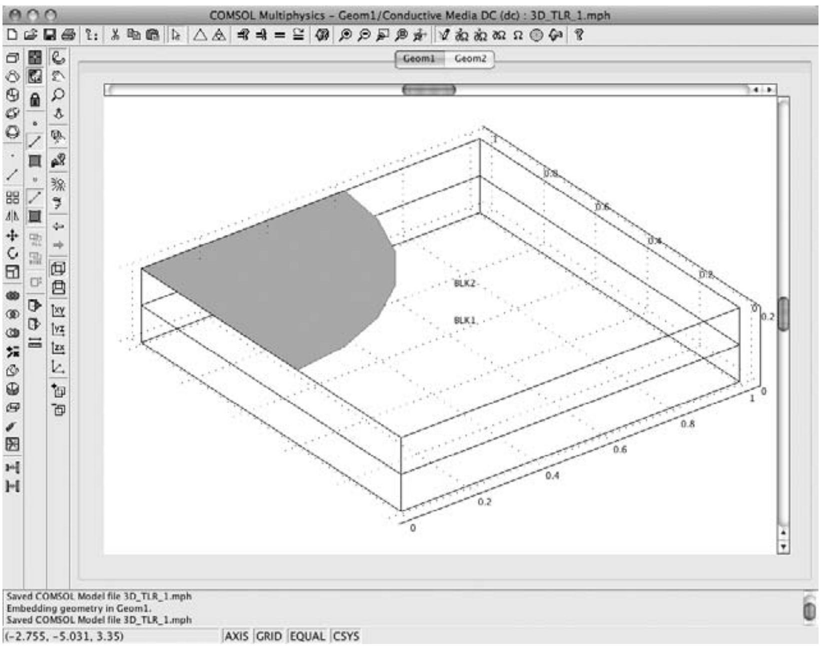


FIGURE 8.17 3D_TLR_1 model EMB1 on top of block BLK2

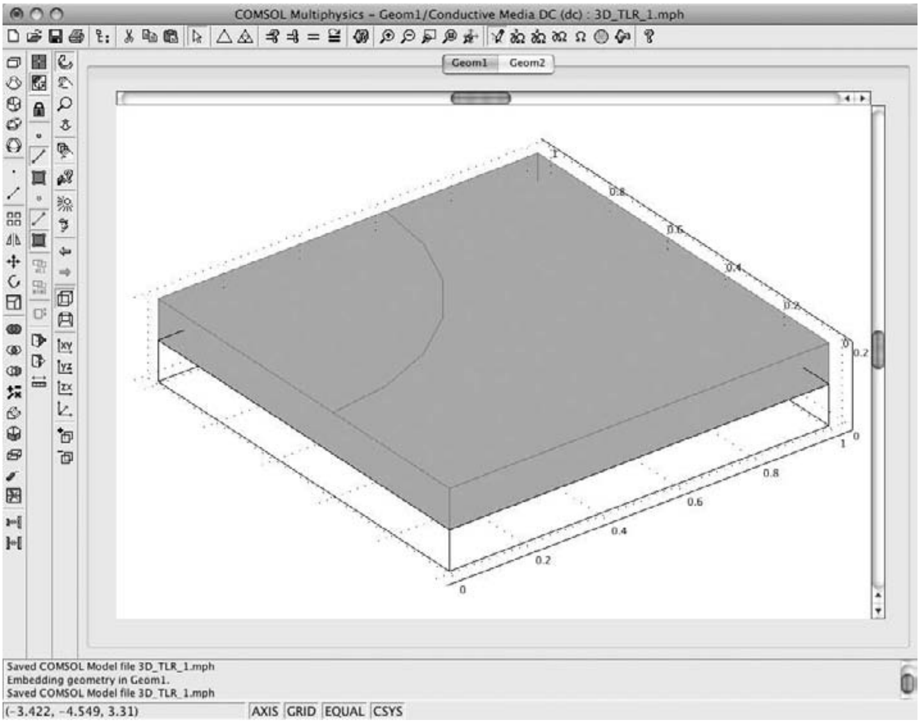


FIGURE 8.18 3D_TLR_1 model with EMB1 and BLK1 selected

Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select subdomains 1 and 2 in the Subdomain selection window (the only available subdomains). In the Subdomain edit windows, enter the information shown in Table 8.2; also see Figure 8.21. Click OK.

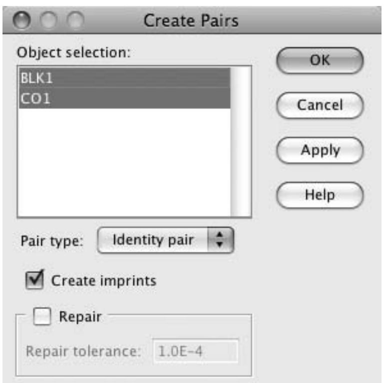


FIGURE 8.19 3D_TLR_1 model Create Pairs edit window

Table 8.2 Subdomain Edit Window

Name	Expression	Description
σ	1	Electrical conductivity

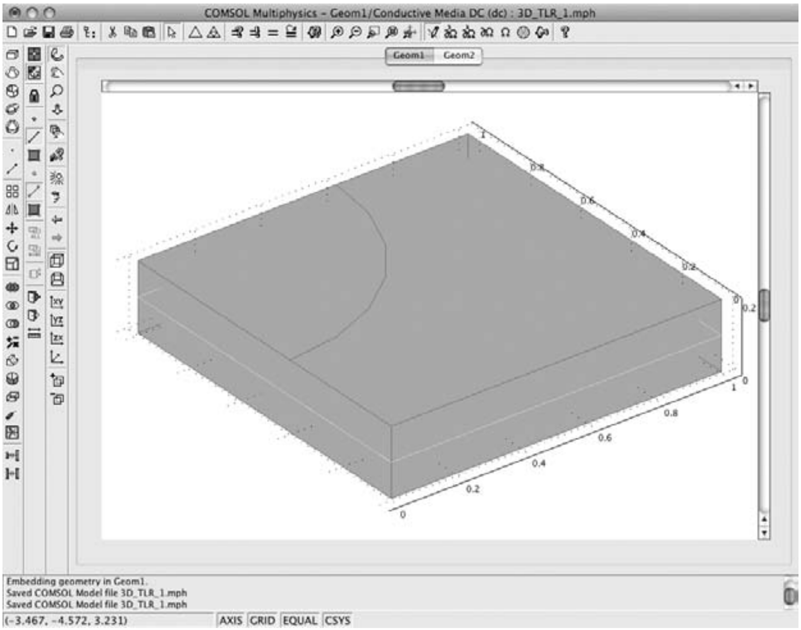


FIGURE 8.20 3D_TLR_1 model identity pair

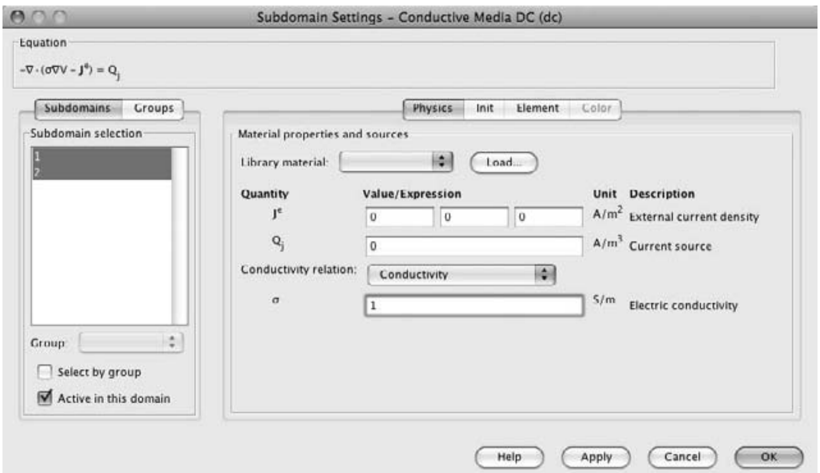


FIGURE 8.21 3D_TLR_1 model Subdomain Settings edit window

Table 8.3 Boundary Settings – Conductive Media DC (dc) Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
1, 2, 4–10, 12, 13	Electric insulation	—	8.22
3	Inward current flow	0.3	8.23
11	Ground	—	8.24

Physics Boundary Settings: Conductive Media DC (dc)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 8.3. See Figures 8.22, 8.23, and 8.24.

Click the Pairs tab. Select "Contact resistance" from the Boundary condition pull-down list. For the indicated quantity, select or enter the given value as shown in Table 8.4; also see Figure 8.25. Click OK.

Table 8.4 Boundary Settings – Conductive Media DC (dc) Edit Window, Pairs

Quantity	Value/Expression	Description
σ	1e-2	Electrical conductivity
d	0.02	Thickness in meters

FIGURE 8.22 3D_TLR_1 model Boundary Settings (1, 2, 4–10, 12, 13) edit window

FIGURE 8.23 3D_TLR_1 model Boundary Settings (3) edit window

NOTE This is the most important step in this model, as it implements the thin layer approximation. By using the identity pair–contact resistance approximation in this model, the modeler has eliminated the necessity of building and using a third domain as the interface layer.

FIGURE 8.24 3D_TLR_1 model Boundary Settings (11) edit window

FIGURE 8.25 3D_TLR_1 model boundary pairs value edit window

Mesh Generation

From the toolbar, select Mesh > Initialize Mesh. See Figure 8.26.

Solving the 3D_TLR_1 Model

Using the menu bar, select Solve > Solve Problem. See Figure 8.27.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Postprocessing and Visualization

The default plot shows a slice plot of the electric potential (V) distribution in volts. To visualize the solution as a boundary plot, the Plot Parameters will need to be modified.

Select Postprocessing > Plot Parameters > General. In the Plot type list, unselect the Slice check box. In the Plot type list, select the Boundary check box.

Click the Boundary tab. Select Conductive Media DC (dc) > Electric potential (V). Unselect the Smooth check box. See Figure 8.28.

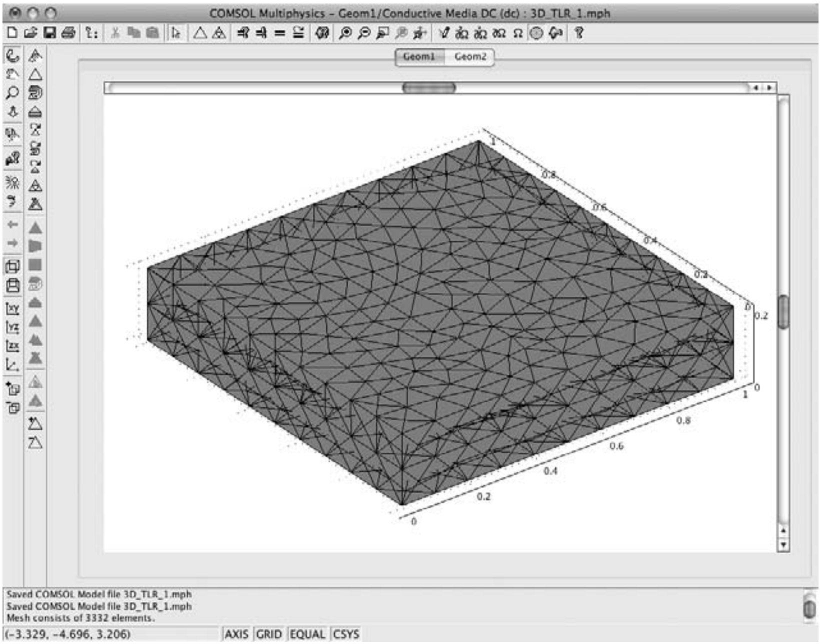


FIGURE 8.26 3D_TLR_1 model mesh

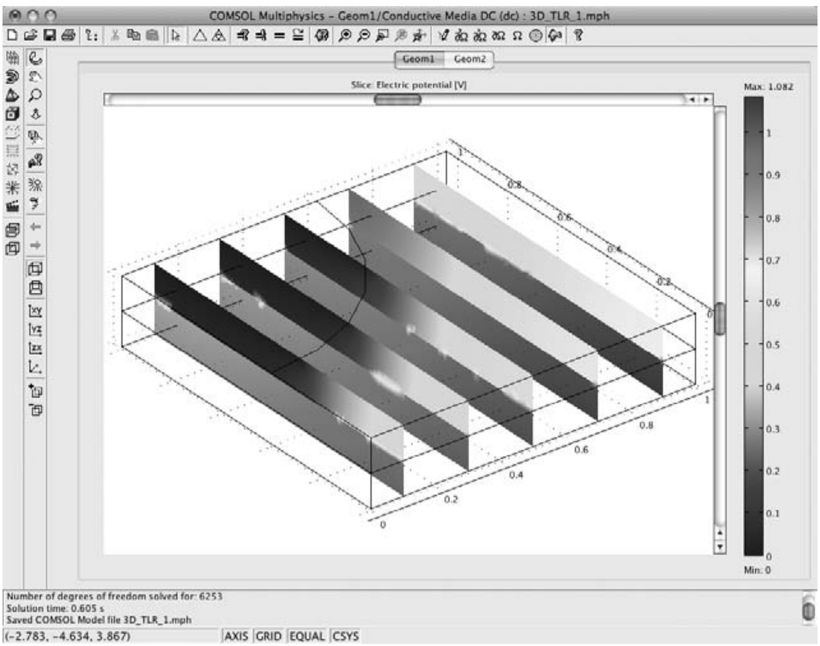


FIGURE 8.27 3D_TLR_1 model solution, default slice plot

FIGURE 8.28 3D_TLR_1 model solution, Plot Parameters edit window, Boundary tab

Click OK. See Figure 8.29.

TLR Voltage Measured Across the Layer

To visualize the voltage across the thin layer resistance, select Postprocessing > Cross-Section Plot Parameters > General. Click the Line/Extrusion plot radio button among the Plot type selection choices. Click the Line/Extrusion tab. Select "Electric Potential" from the y-axis data pull-down list. Click the Expression radio button. Click the Expression button and enter z in the edit window. See Figure 8.30.

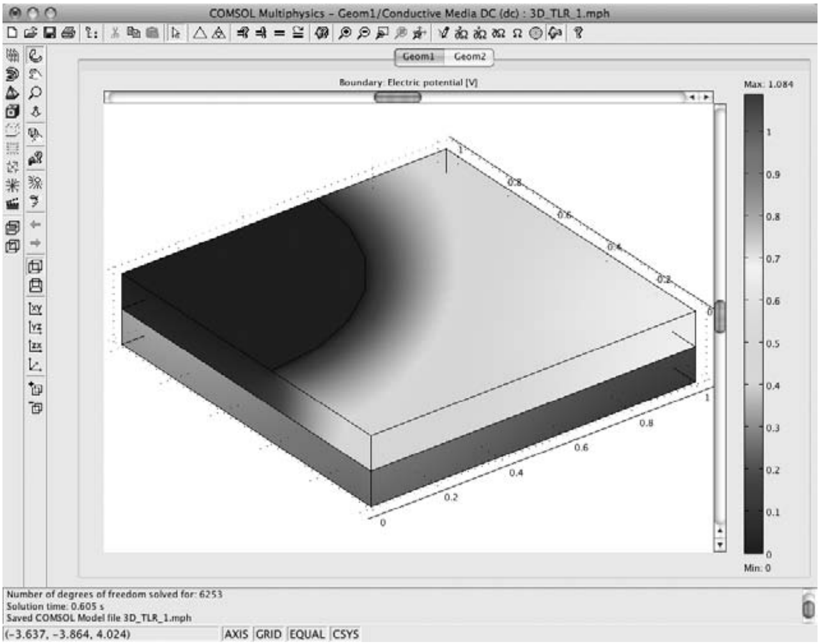


FIGURE 8.29 3D_TLR_1 model solution, boundary plot

Click OK. For the cross-section line data, select or enter the given value as shown in Table 8.5. See Figure 8.31.

Click OK. See Figure 8.32.

3D Thin Layer Resistance Model, Thin Layer Approximation: Summary and Conclusions

The 3D thin layer resistance model, thin layer approximation, has now been built and solved. This model employs the thin layer approximation to solve a model by replacing the center domain with a contact resistance identity pair. Such an approximation

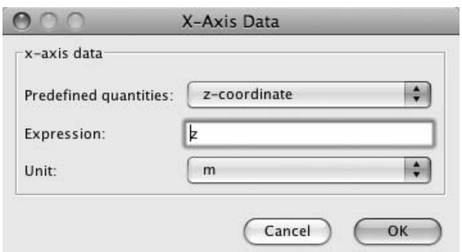


FIGURE 8.30 3D_TLR_1 model, X-Axis Data edit window

Table 8.5 Cross-Section Line Data Parameters

Quantity	Value/Expression
x0	0.5
y0	0.5
z0	0.0
x1	0.5
y1	0.5
z1	0.2

has a broad applicability: It can be used in any problem in which flow is described by the divergence of a gradient flux (i.e., diffusion, heat conduction, flow through porous media under Darcy's law).

The application of the thin layer approximation is especially valuable to the modeler when the differences in domain thickness are so great that the mesh generator fails to properly mesh the model or creates more elements than the modeling platform can

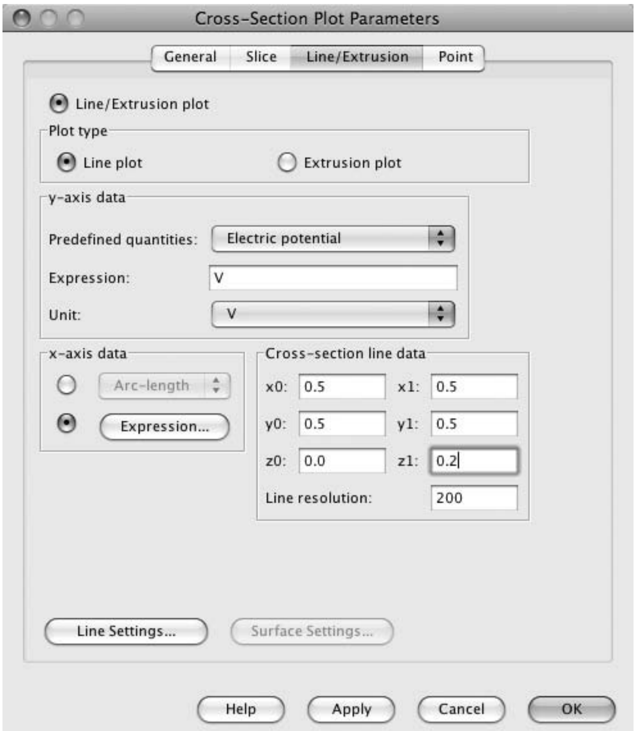


FIGURE 8.31 3D_TLR_1 model, Cross-Section Plot Parameters edit window

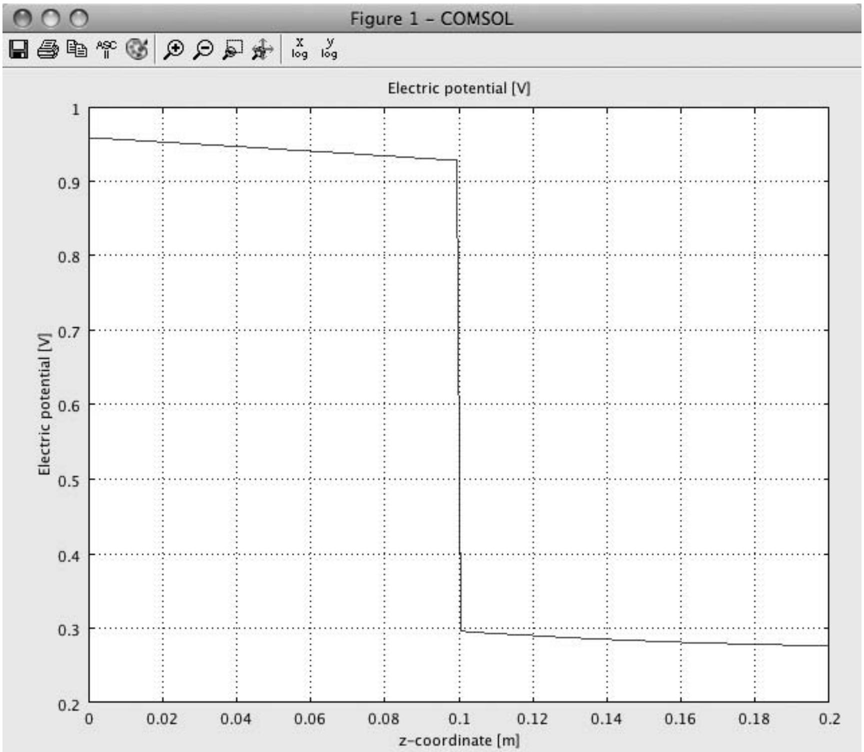


FIGURE 8.32 3D_TLR_1 model, cross-section electric potential plot

handle (the "run out of memory" problem). In those cases, this approximation may enable a model to be solved that would otherwise fail.

3D Thin Layer Resistance Model: Thin Layer Subdomain

The following numerical solution model (3D_TLR_2 model) is similar to the previous model. However, in this case, the center layer is a full domain.

To start building the 3D_TLR_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select COMSOL Multiphysics > Electromagnetics > Conductive Media DC. See Figure 8.33. Click OK.

Geometry Modeling

Using the menu bar, select Draw > Block. In the Block edit window, enter the information shown in Table 8.6. Click OK after filling in the parameters of each separate block in the Block edit window. See Figures 8.34, 8.35, and 8.36.

Table 8.6 Geometry Components

Name	Style	Base	Length (X, Y, Z)	Axis Base Point (X, Y, Z)	Figure Number
BLK1	Solid	Corner	(1, 1, 0.1)	(0, 0, 0)	8.34
BLK2	Solid	Corner	(1, 1, 0.02)	(0, 0, 0.1)	8.35
BLK3	Solid	Corner	(1, 1, 0.1)	(0, 0, 0.12)	8.36

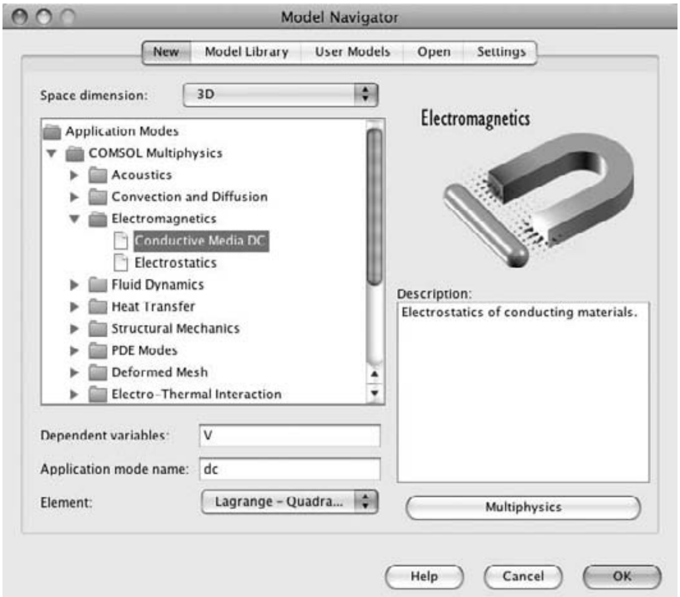


FIGURE 8.33 3D_TLR_2 Model Navigator setup

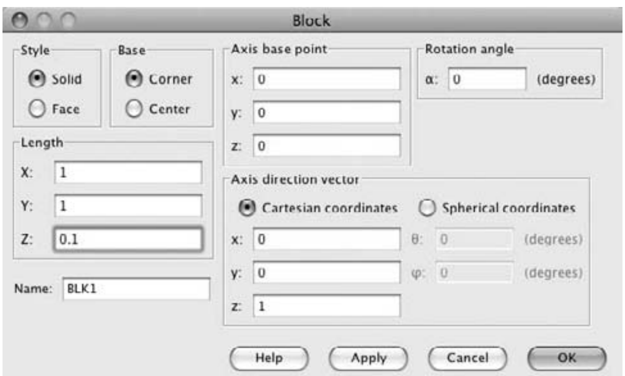


FIGURE 8.34 3D_TLR_2 model BLK1 edit window

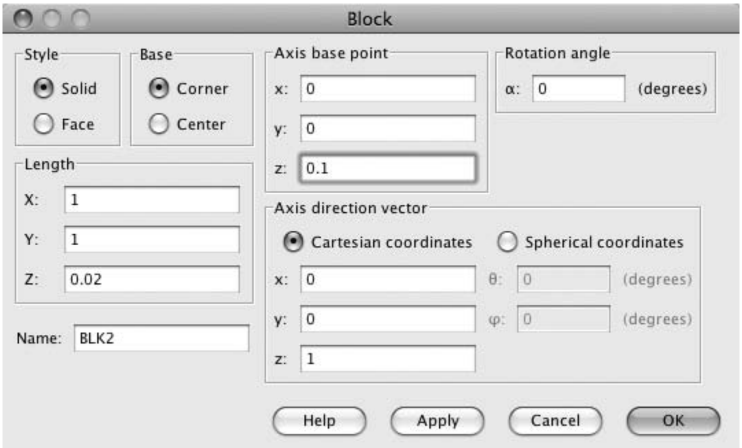


FIGURE 8.35 3D_TLR_2 model BLK2 edit window

Click the Zoom Extents button. See Figure 8.37.

Select File > Save As. Enter 3D_TLR_2 in the Save As edit window. See Figure 8.38. Click the Save button.

Using the menu bar, select Draw > Work-Plane Settings. See Figure 8.39.

NOTE As noted earlier, the use of the Work-Plane Settings is intended to make specific 2D planes available to the modeler to facilitate the creation of essentially 2D objects in the 3D geometry.

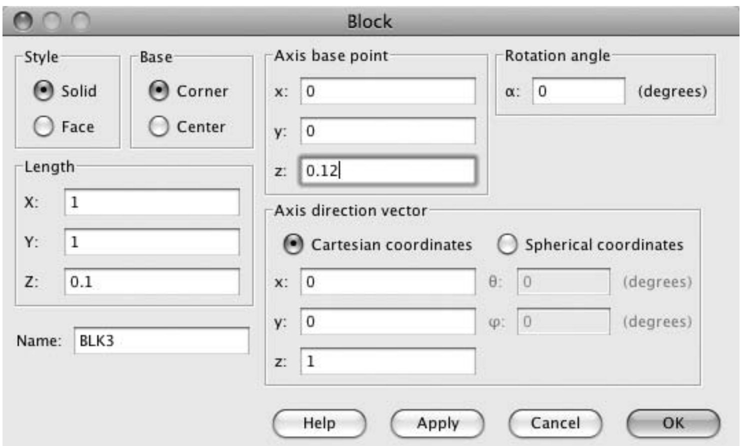


FIGURE 8.36 3D_TLR_2 model BLK3 edit window

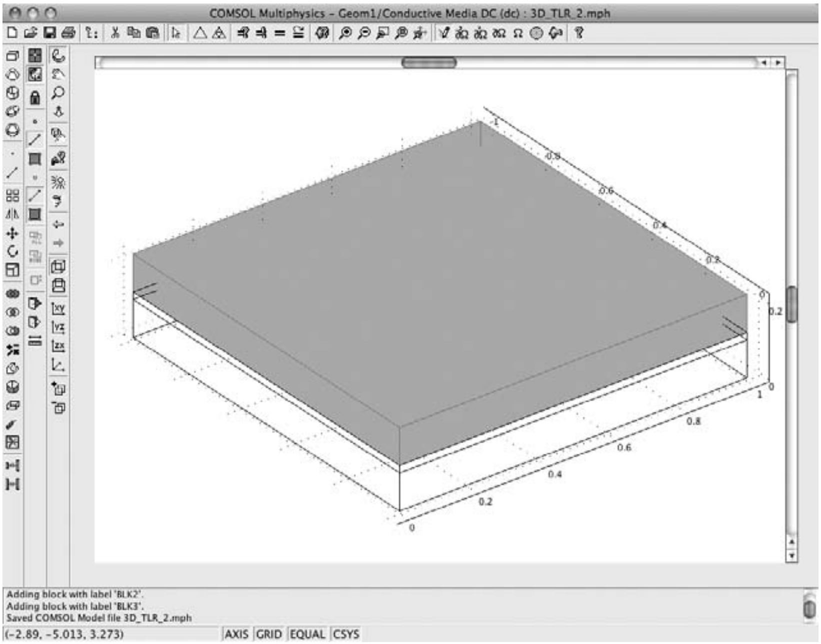


FIGURE 8.37 3D_TLR_2 model

Click OK, using the default settings. Click the Zoom Extents button. See Figure 8.40.

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.6, set the base to Center, and set x equal to 0 and y equal to 1. See Figure 8.41. Click OK.

FIGURE 8.38 3D_TLR_2 model Save As edit window

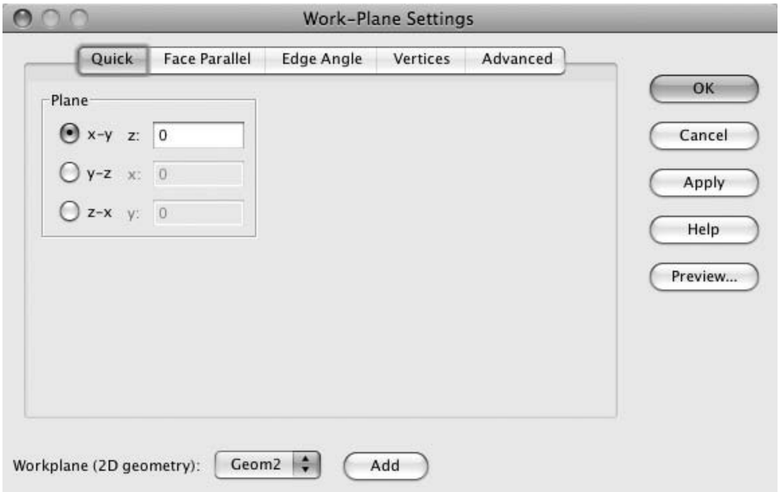


FIGURE 8.39 3D_TLR_2 model Work-Plane Settings edit window

FIGURE 8.40 3D_TLR_2 model Geom2 work-plane

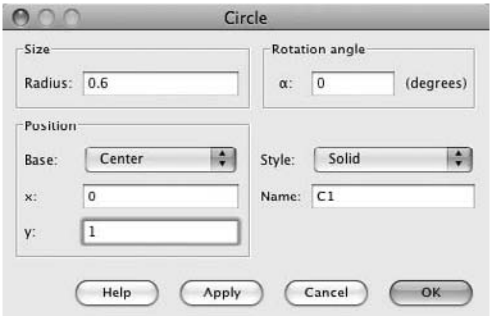


FIGURE 8.41 3D_TLR_2 model Circle edit window

Using the menu bar, select Draw > Specify Objects > Square. Enter a width of 1, set the base as Corner, and set x equal to 0 and y equal to 0. See Figure 8.42. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter C1* SQ1 in the Set formula edit window. See Figure 8.43.

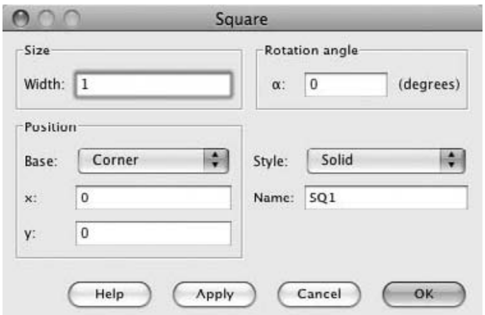


FIGURE 8.42 3D_TLR_2 model Square edit window

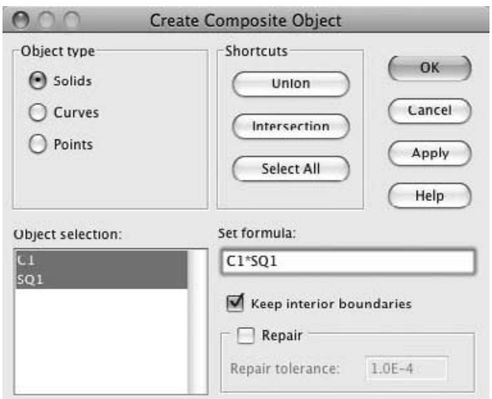


FIGURE 8.43 3D_TLR_2 model Create Composite Object edit window

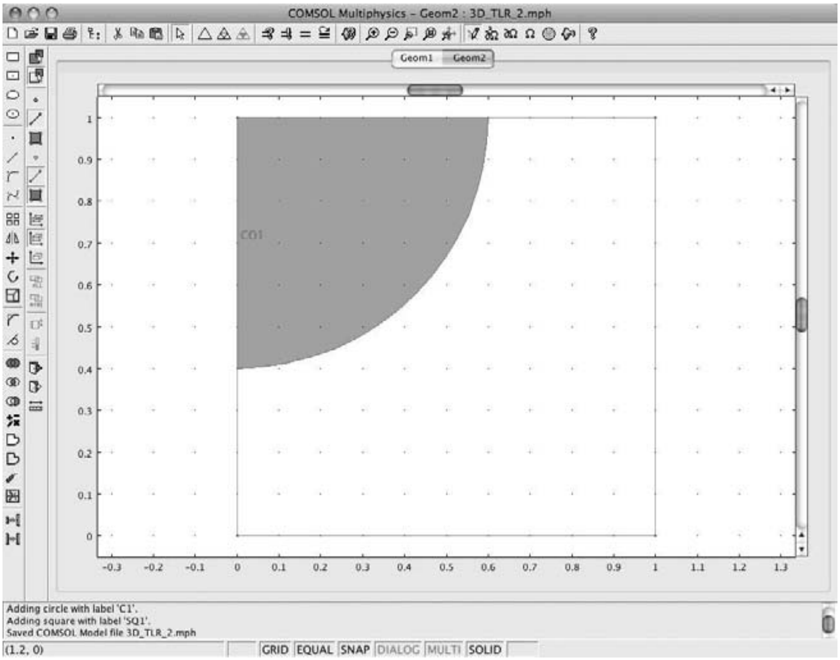


FIGURE 8.44 3D_TLR_2 model CO1 (intersection of circle and square)

NOTE The formula $X*Y$ creates the intersection of X and Y.

Click OK. See Figure 8.44.

Using the menu bar, select Draw > Embed. See Figure 8.45. Click OK.

As is obvious, the quarter-circle electrode needs to be moved to the upper surface of the upper block. Using the menu bar, select Draw > Modify > Move. Enter x = 0, y = 0, and z = 0.22. See Figure 8.46.

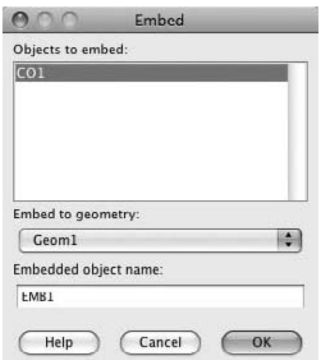


FIGURE 8.45 3D_TLR_2 model Embed edit window

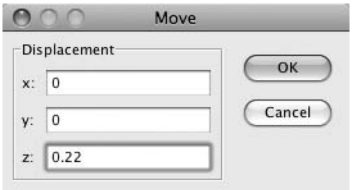


FIGURE 8.46 3D_TLR_2 model Move edit window

Click OK. See Figure 8.47.

Select EMB1 and BLK3 (click on EMB1 and then shift-click on BLK3). See Figure 8.48. Using the menu bar, select Draw > Coerce To > Solid.

Physics Subdomain Settings: Conductive Media DC (dc)

Having established the geometry for the 3D_TLR_2 model of three blocks and an electrode, the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. In the Subdomain edit windows, enter the information shown in Table 8.7. Click OK. See Figures 8.49 and 8.50.

FIGURE 8.47 3D_TLR_2 model EMB1 on top of block BLK3

Table 8.7 Subdomain Edit Window

Subdomain	Name	Expression	Description	Figure Number
1, 3	σ	1	Electrical conductivity	8.49
2	σ	1e-2	Electrical conductivity	8.50

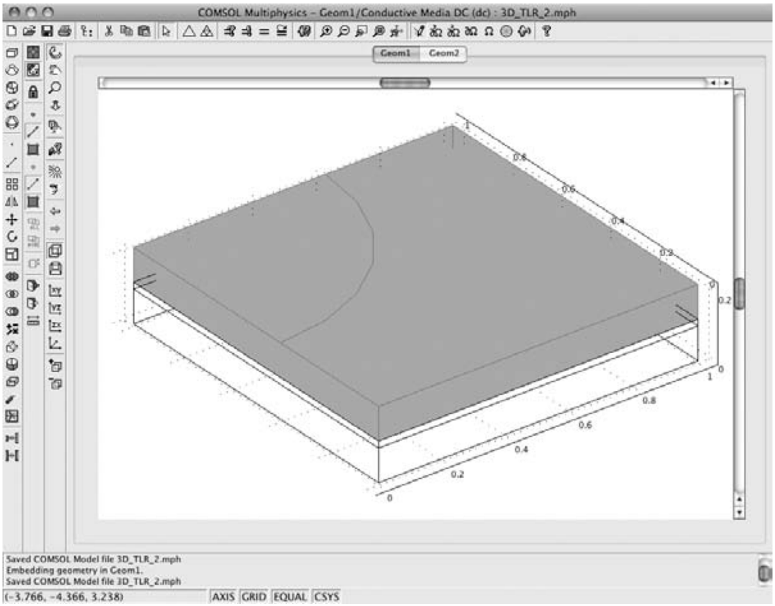


FIGURE 8.48 3D_TLR_2 model with EMB1 and BLK3 selected

Physics Boundary Settings: Conductive Media DC (dc)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 8.8. Click OK. See Figures 8.51, 8.52, and 8.53.

Table 8.8 Boundary Settings – Conductive Media DC (dc) Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
1, 2, 4, 5, 7, 8, 10, 12–17	Electric insulation	—	8.51
3	Inward current flow	0.3	8.52
11	Ground	—	8.53

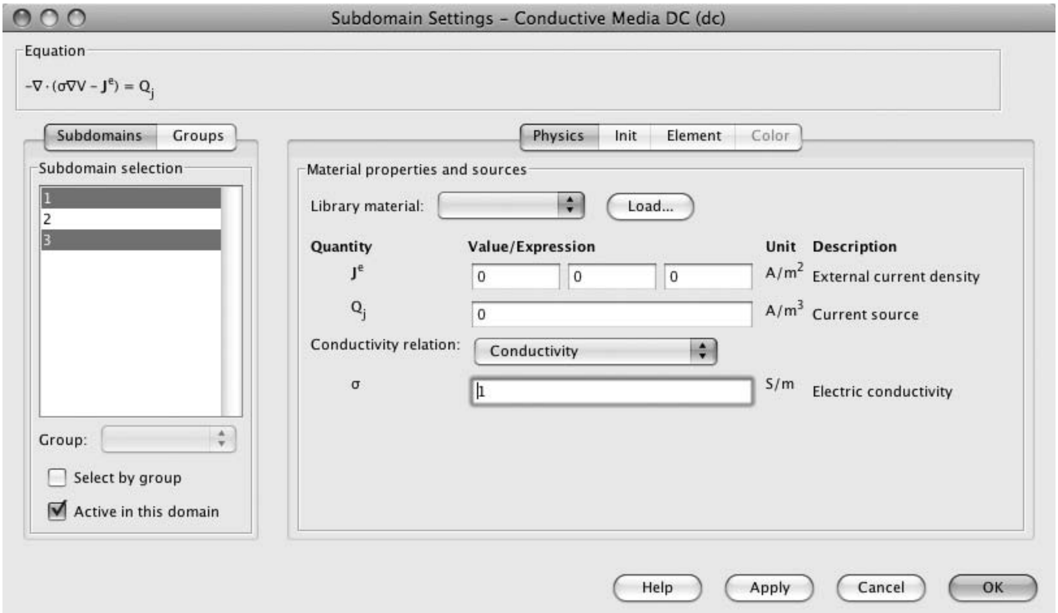


FIGURE 8.49 3D_TLR_2 model Subdomain Settings (1, 3) edit window

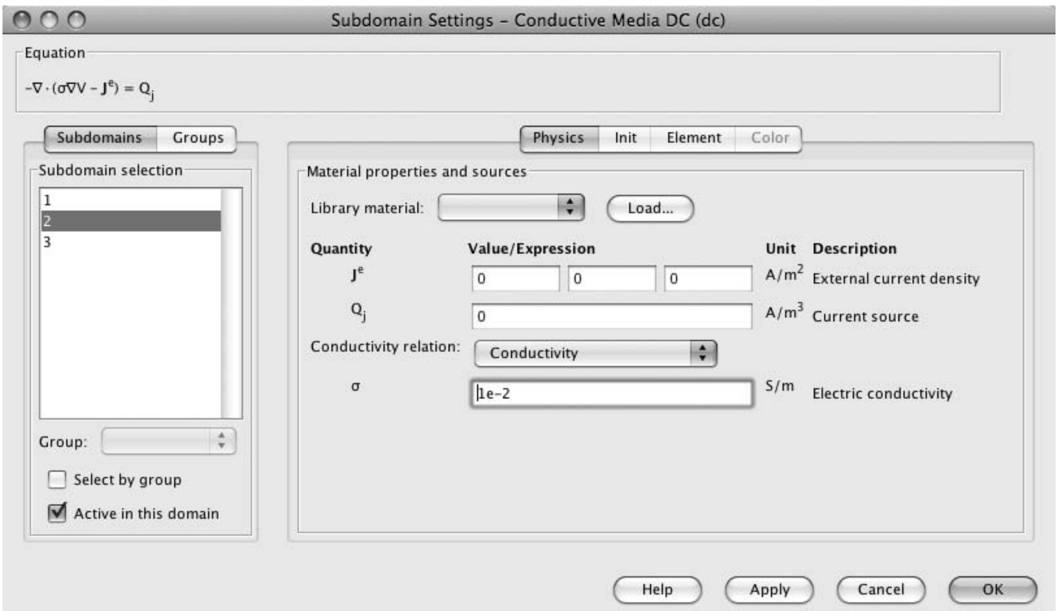


FIGURE 8.50 3D_TLR_2 model Subdomain Settings (2) edit window

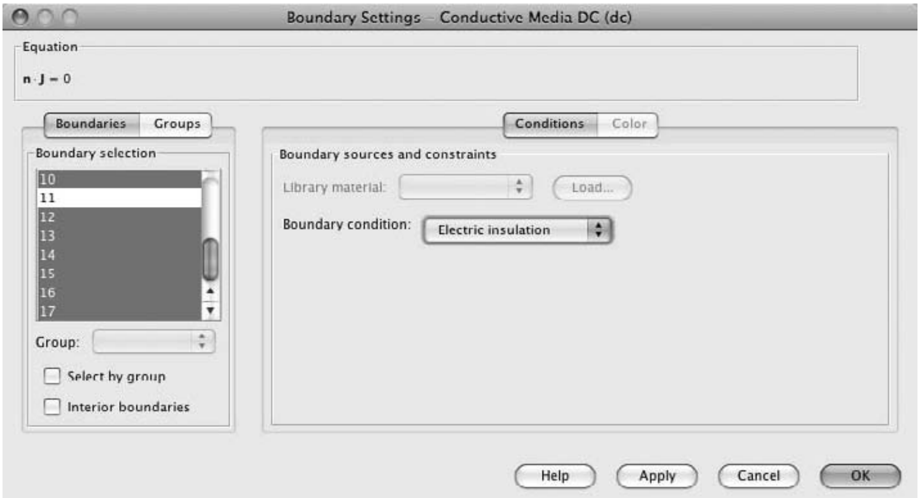


FIGURE 8.51 3D_TLR_2 model Boundary Settings (1, 2, 4, 5, 7, 8, 10, 12–17) edit window

NOTE In this model, the thin layer approximation has been replaced by subdomain 2.

Mesh Generation

From the toolbar, select Mesh > Free Mesh Parameters > Subdomain. Select subdomain 2. Enter 0.02 in the Maximum element size edit window. See Figure 8.54. Click the Mesh Selected button.

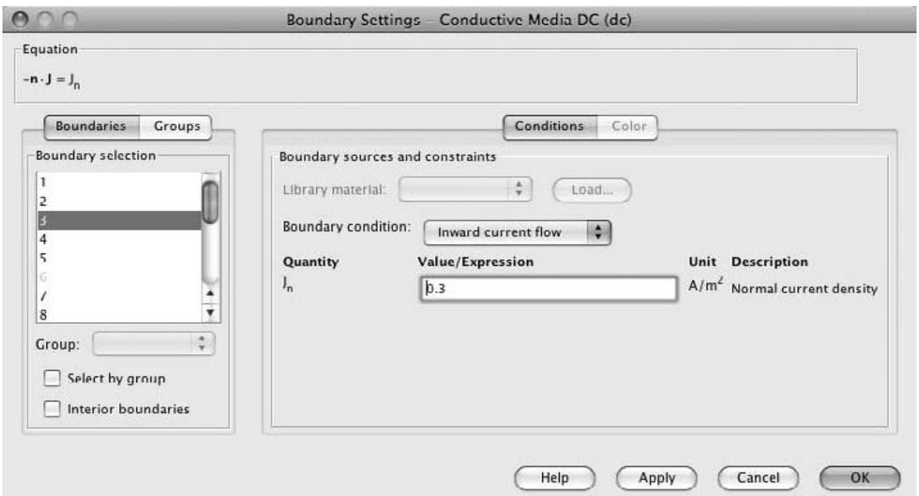


FIGURE 8.52 3D_TLR_2 model Boundary Settings (3) edit window

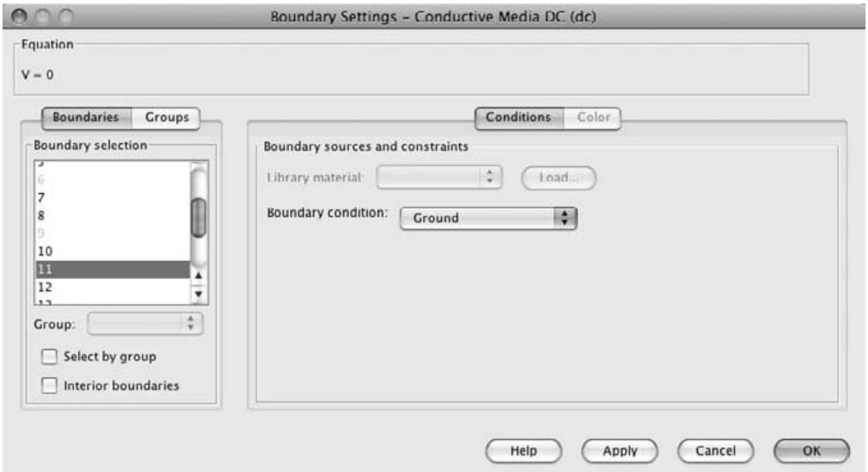


FIGURE 8.53 3D_TLR_2 model Boundary Settings (11) edit window

Click the Select Remaining button. (Subdomains 1 and 3 will be highlighted.) Click the Mesh Selected button. Click OK. See Figure 8.55.

Solving the 3D_TLR_2 Model

Using the menu bar, select Solve > Solve Problem. See Figure 8.56.

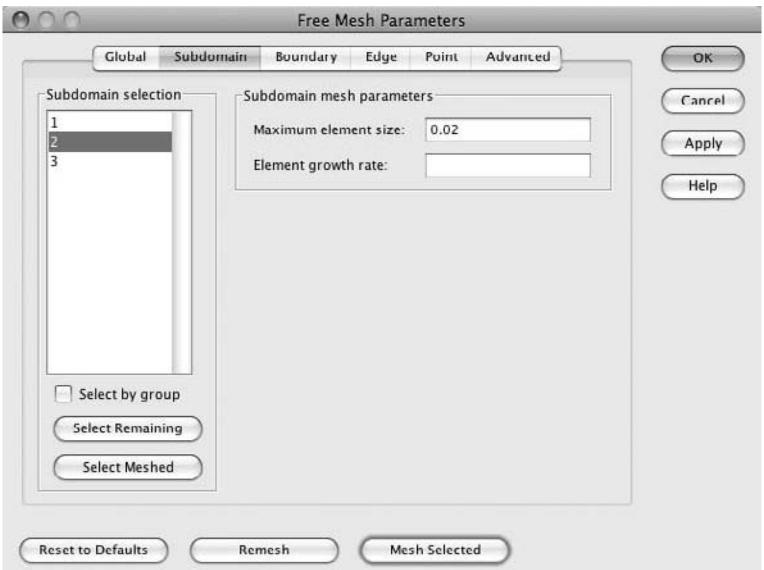


FIGURE 8.54 3D_TLR_2 model Free Mesh Parameters edit window

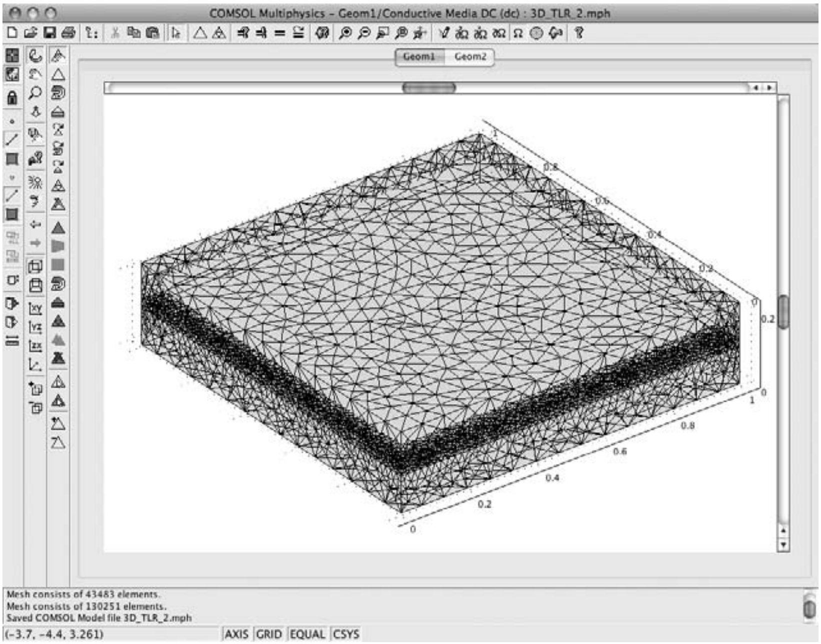


FIGURE 8.55 3D_TLR_2 model mesh

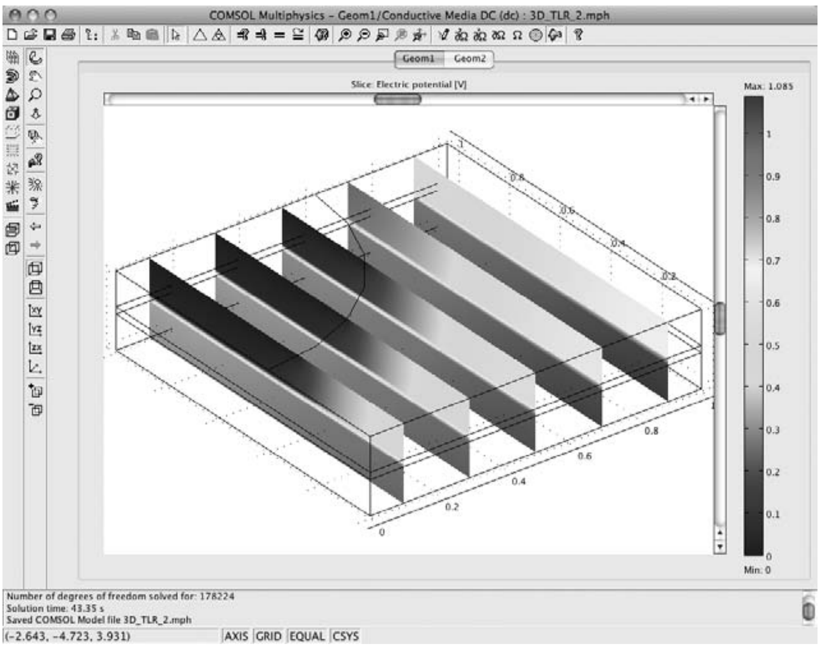


FIGURE 8.56 3D_TLR_2 model solution, default slice plot

FIGURE 8.57 3D_TLR_2 model Plot Parameters edit window, General tab

---

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Postprocessing and Visualization

The default plot shows a slice plot of the electric potential (V) distribution in volts. To visualize the solution as a boundary plot, the Plot Parameters will need to be modified.

Select Postprocessing > Plot Parameters > General. In the Plot type list, unselect the Slice check box. In the Plot type list, select the Boundary check box. See Figure 8.57.

FIGURE 8.58 3D_TLR_2 model solution Plot Parameters edit window, Boundary tab

Click the Boundary tab. Select Conductive Media DC (dc) > Electric potential (V). Unselect the Smooth check box. See Figure 8.58.

Click OK. See Figure 8.59.

TLR Voltage Measured Across the Layer

To visualize the voltage across the thin layer resistance (subdomain 2), select Postprocessing > Cross-Section Plot Parameters > General. Click the Line/Extrusion plot radio button among the Plot type selection choices.

Click the Line/Extrusion tab. Select "Electric Potential" from the y-axis data pull-down list. Click the Expression radio button. Click the Expression button and enter *z* in the edit window. See Figure 8.60. Click OK.

FIGURE 8.59 3D_TLR_2 model solution, boundary plot

For the cross-section line data, select or enter the given value as shown in Table 8.9. See Figure 8.61.

Click OK. See Figure 8.62.

3D Thin Layer Resistance Models: Summary and Conclusions

The 3D thin layer resistance model, thin layer approximation, and the 3D thin layer resistance model, thin layer subdomain, have now been built and solved. A direct

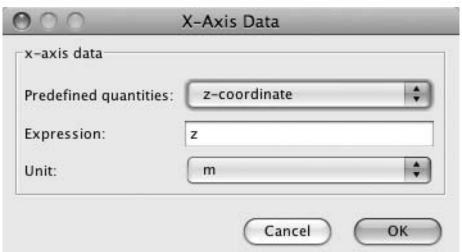


FIGURE 8.60 3D_TLR_2 model X-Axis Data edit window

Table 8.9 Cross-Section Line Data Parameters

Quantity	Value/Expression
x0	0.5
y0	0.5
z0	0.0
x1	0.5
y1	0.5
z1	0.2

comparison can be made of the model solutions by comparing the results obtained from the cross-section plots. See Figures 8.63 and 8.64.

As can be seen from the examination of the plots, the only substantial difference between the two solutions is the electrical potential difference across subdomain 2. Thus the modeler can choose the implementation that best suits his or her system and time constraints, without suffering excessive inaccuracies based on the approximation method.

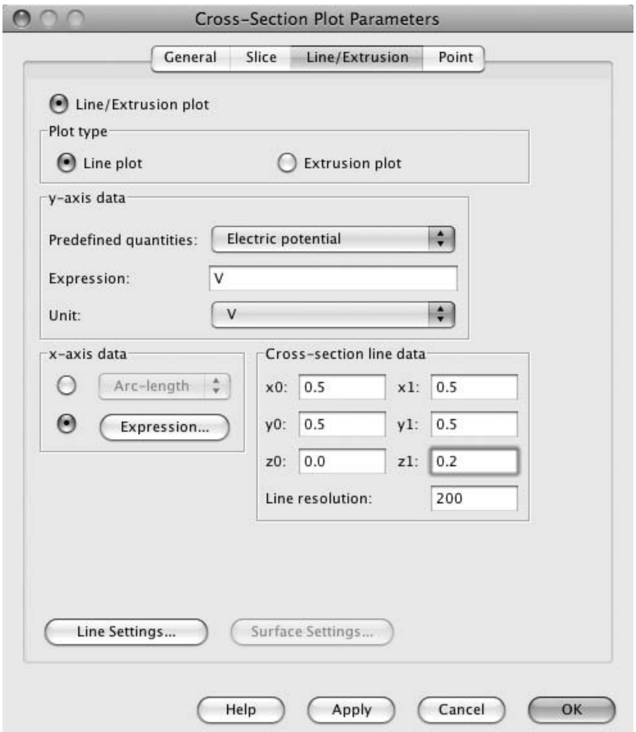


FIGURE 8.61 3D_TLR_2 model Cross-Section Plot Parameters edit window

FIGURE 8.62 3D_TLR_2 model, cross-section electric potential plot

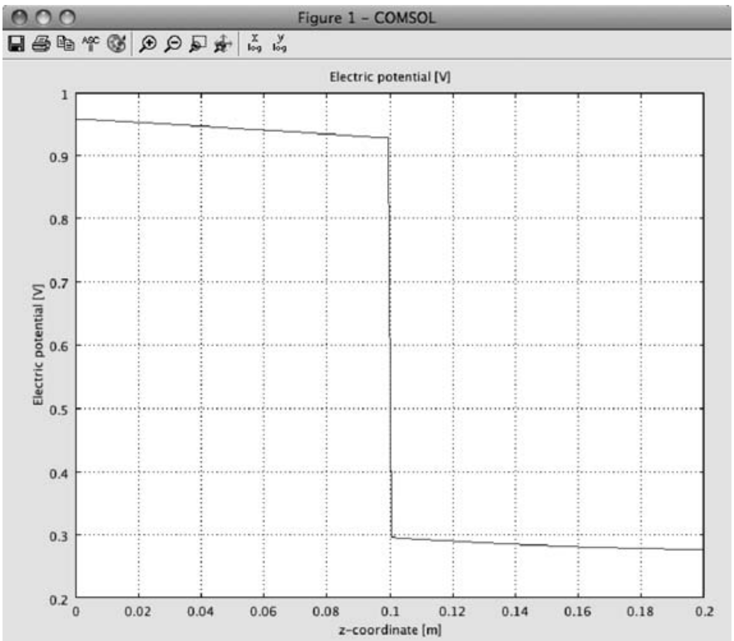


FIGURE 8.63 3D_TLR_1 model, cross-section electric potential plot, thin layer approximation

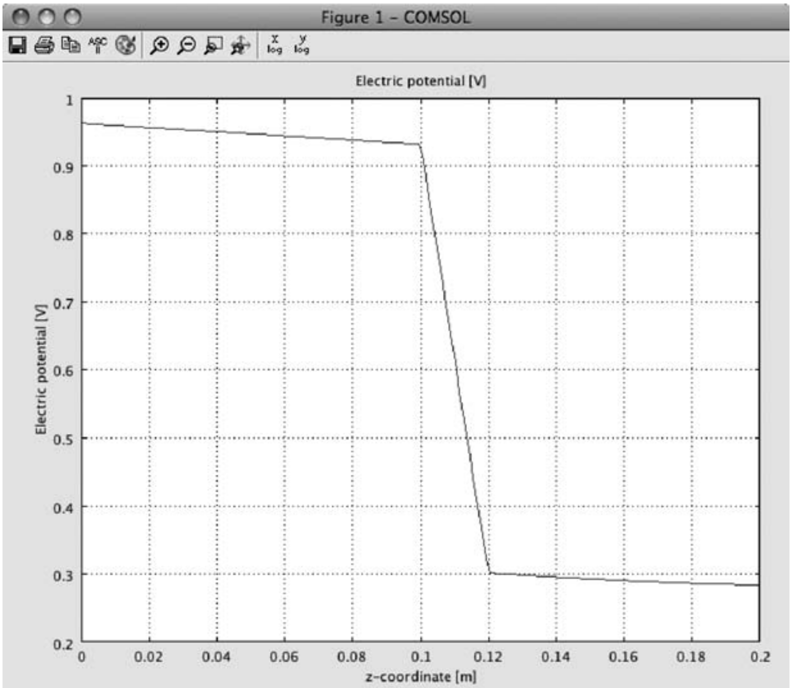


FIGURE 8.64 3D_TLR_2 model, cross-section electric potential plot, thin layer subdomain

Electrostatic Modeling Basics

The study of static electricity, a well-known and widely observed phenomenon, has a long history. Thales of Miletus[7] recorded the first known scientific observations using amber in approximately the sixth century BC. Additional serious documented scientific work on static electricity did not occur until Otto von Guericke[8] invented the first electrostatic generator[9] around 1663.

The physics of static electricity was not well understood until the work of Charles-Augustin de Coulomb,[10] Johann Carl Friedrich Gauss,[11] and others[12] explored electrostatics and mathematics of physics in the late 1700s to early 1800s AD. Based on that work, the electrostatic scalar potential (V) is known to be related to the electric field vector ($\mathbf{E}$) as follows:

$$\mathbf{E} = -\nabla V \tag{8.13}$$

where $\mathbf{E}$ = electric field vector (V/m)
∇ = divergence operator (1/m)
V = scalar electric potential (V)

NOTE The divergence operator is as follows:

$$\nabla = \hat{i}\frac{\partial}{\partial x} + \hat{j}\frac{\partial}{\partial y} + \hat{k}\frac{\partial}{\partial z}$$

where i, j, k are the unit vectors in the x, y, z directions, respectively.

Using Gauss's law,[13]

$$\nabla \cdot (\varepsilon \mathbf{E}) = \rho \tag{8.14}$$

where $\mathbf{E}$ = electric field vector (V/m)
∇ = divergence operator (1/m)
ε = permittivity
ρ = space charge density

Substituting for $\mathbf{E}$ gives

$$-\nabla \cdot (\varepsilon \nabla V) = \rho \tag{8.15}$$

and

$$-\nabla \cdot (\varepsilon \nabla V) = -\varepsilon \nabla^2 V = \rho \tag{8.16}$$

where ∇^2 is the Laplacian operator.

NOTE The Laplacian operator[14,15] is as follows:

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2}$$

It is a differential operator, shown above in the scalar form, named after Pierre-Simon de Laplace.[16] The Laplacian operator is widely employed in the physics of electromagnetics, wave propagation, heat flow, fluid flow, and quantum mechanics, to name a few areas.

A large body of subsequent work has led to both scientific and engineering applications (e.g., ranging from X-ray tubes and particle accelerators to paint sprayers and dust precipitators).

The 3D electrostatic potential models presented in this section are examples of methods that can be used by the modeler to explore electrostatic potentials in different geometric configurations.

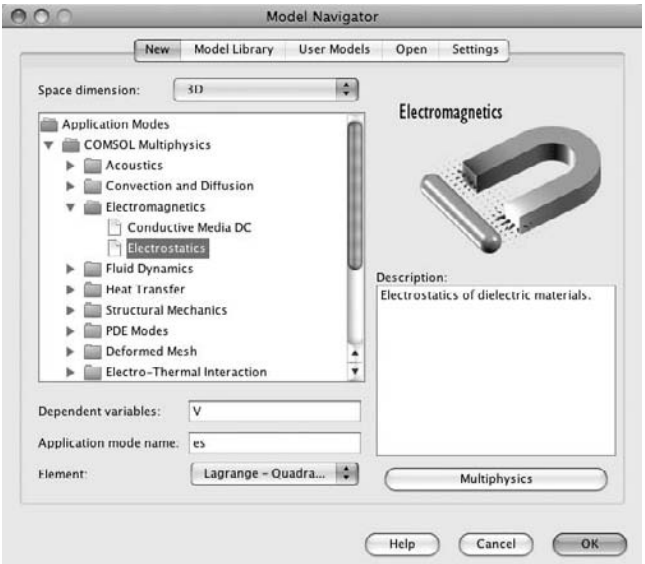


FIGURE 8.65 3D_ESP_1 Model Navigator setup

3D Electrostatic Potential Between Two Cylinders

The following numerical solution model (3D_ESP_1 model) is derived from a model that was originally developed by COMSOL as a Multiphysics Electromagnetics demonstration model. That model was developed for distribution with the Multiphysics software as a COMSOL Multiphysics Model Library.

To start building the 3D_ESP_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select COMSOL Multiphysics > Electromagnetics > Electrostatics. See Figure 8.65. Click OK.

Geometry Modeling

Using the menu bar, select Draw > Sphere. In the Sphere edit window, enter the information shown in Table 8.10. Click OK after filling in the parameters of each separate solid in the appropriate edit window. See Figures 8.66, 8.67, and 8.68.

Table 8.10 Geometry Components

Solid	Name	Style	Radius	Height	Axis Base Point (X, Y, Z)	Axis Direction Vector (X, Y, Z)	Figure Number
Sphere	SPH1	Solid	2		(0, 0, 0)	(0, 0, 1)	8.66
Cylinder	CYL1	Solid	0.1	0.4	(−0.4, 0, −0.2)	(0, 0, 1)	8.67
Cylinder	CYL2	Solid	0.1	0.4	(0.4, 0, −0.2)	(0, 0, 1)	8.68

FIGURE 8.66 3D_ESP_1 model Sphere SPH1 edit window

FIGURE 8.67 3D_ESP_1 model Cylinder CYL1 edit window

FIGURE 8.68 3D_ESP_1 model Cylinder CYL2 edit window

FIGURE 8.69 3D_ESP_1 model Save As edit window

Click the Zoom Extents button. Select File > Save As. Enter 3D_ESP_1.mph in the Save As edit window. See Figure 8.69. Click the Save button.

Using the menu bar, select Draw > Create Composite Object. Enter SPH1-CYL1-CYL2 in the Set formula edit window. See Figure 8.70. Click OK.

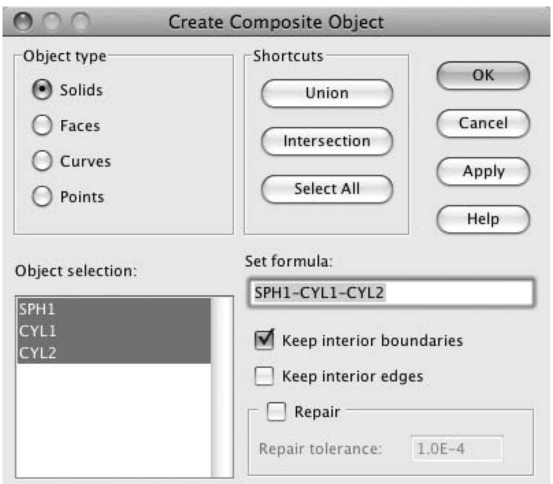


FIGURE 8.70 3D_ESP_1 model Create Composite Object edit window

FIGURE 8.71 3D_ESP_1 model Subdomain Settings edit window

Physics Subdomain Settings: Electrostatics (es)

Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 (the only subdomain available). Click the radio button $D = \varepsilon_0\varepsilon_r E$. Enter 0 in the Space charge density (ρ) edit window. Enter 1 in the Relative permittivity (ε_r) edit window. See Figure 8.71. Click OK.

NOTE The numerical value of the permittivity for free space (ε_0) in SI units is $8.854*10^{12}$ F/m. That value is the default value for permittivity incorporated into the COMSOL Multiphysics software. The permittivity of a material is the product of ε_0 and ε_r, which in this case is 1.

Physics Boundary Settings: Electrostatics (es)

Using the menu bar, select Physics > Boundary Settings. Select or enter the settings as indicated in Table 8.11. Click OK. See Figures 8.72, 8.73, and 8.74.

Table 8.11 Boundary Settings

Boundary	Settings	Value	Figure Number
1–4, 11–14	Zero charge/symmetry	—	8.72
5–10	Electric potential	1V	8.73
15–20	Electric potential	−1V	8.74

FIGURE 8.72 3D_ESP_1 model Boundary Settings (1–4, 11–14) edit window

Mesh Generation

Using the menu bar, select Mesh > Free Mesh Parameters. Select "Coarser" from the Predefined mesh sizes pull-down list. See Figure 8.75. Click OK.

Using the menu bar, select Mesh > Initialize Mesh. See Figure 8.76.

FIGURE 8.73 3D_ESP_1 model Boundary Settings (5–10) edit window

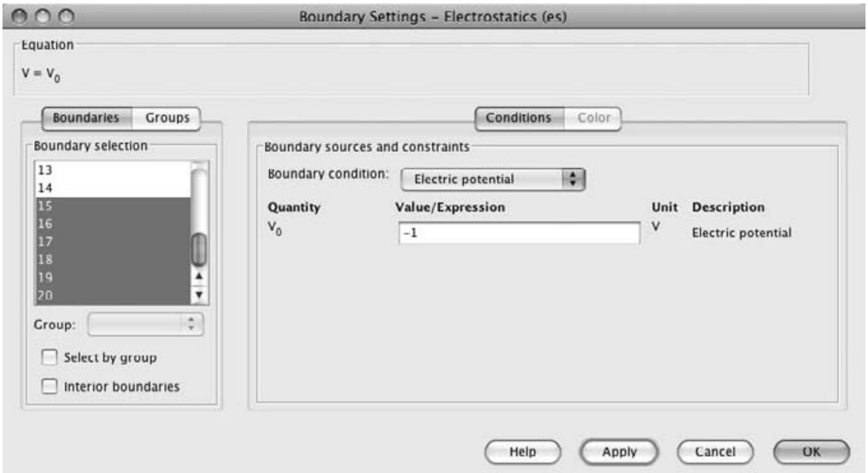


FIGURE 8.74 3D_ESP_1 model Boundary Settings (15–20) edit window

Solving the 3D_ESP_1 Model

NOTE Electrostatics problems can be complex and difficult. The modeler can, of course, accept the COMSOL software default settings. However, in this case, depending on the modeler's platform, it will probably be best to choose an iterative solver (GMRES) and an appropriate preconditioner (Algebraic multigrid). These choices will reduce both the memory required and the time to solution.

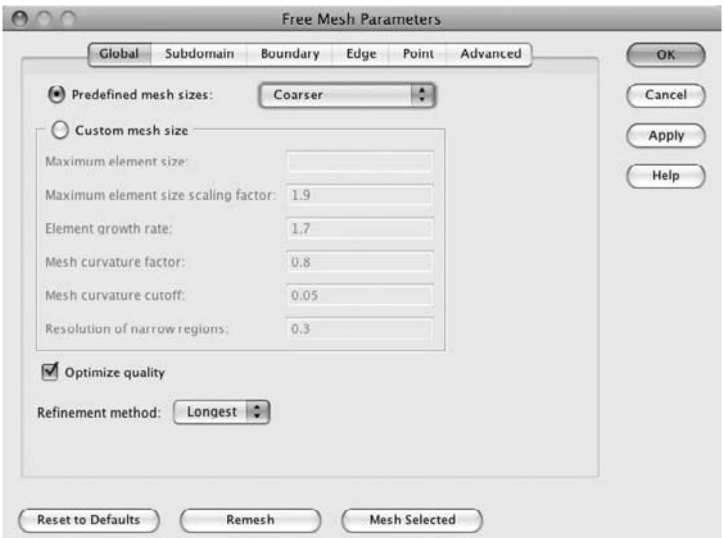


FIGURE 8.75 3D_ESP_1 model Free Mesh Parameters edit window

FIGURE 8.76 3D_ESP_1 model mesh

Select Solve > Solver Parameters. Select "GMRES" from the Linear system solver pull-down list. Select "Algebraic multigrid" from the Preconditioner pull-down list. See Figure 8.77.

Click OK. Select Solve > Solve Problem.

Postprocessing and Visualization

The default solution plot is the slice plot. See Figure 8.78.

Using the menu bar, select Postprocessing > Plot Parameters > General. Uncheck the Slice check box, and check the Boundary and Streamline check boxes. See Figure 8.79.

Click the Boundary tab. Click the Apply button.

Click the Streamline tab. Enter 30 in the Number of start points window. Click OK.

To see the streamline plot, the modeler will need to suppress the boundaries of the sphere. Using the menu bar, select Options > Suppress > Suppress Boundaries. Select boundaries 1–4 and 11–14. See Figure 8.80. Click OK.

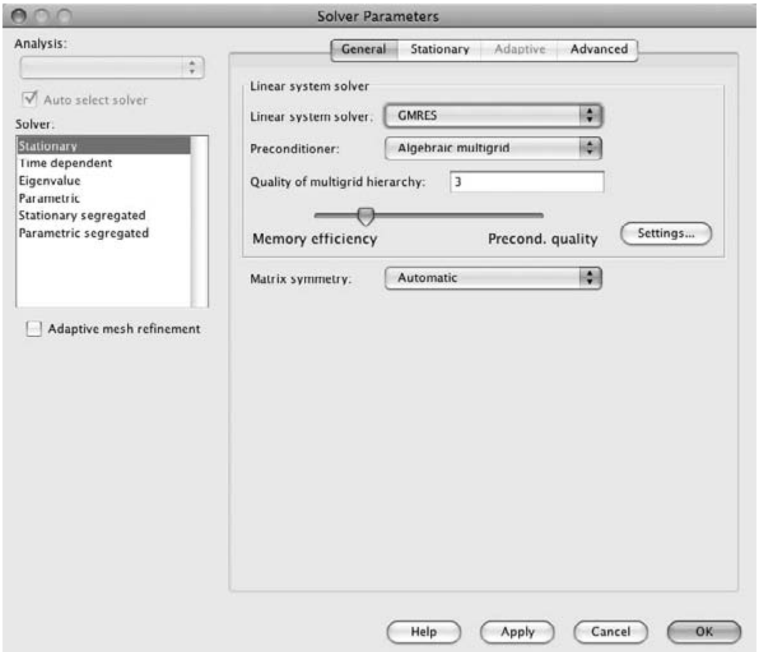


FIGURE 8.77 3D_ESP_1 model Solver Parameters edit window

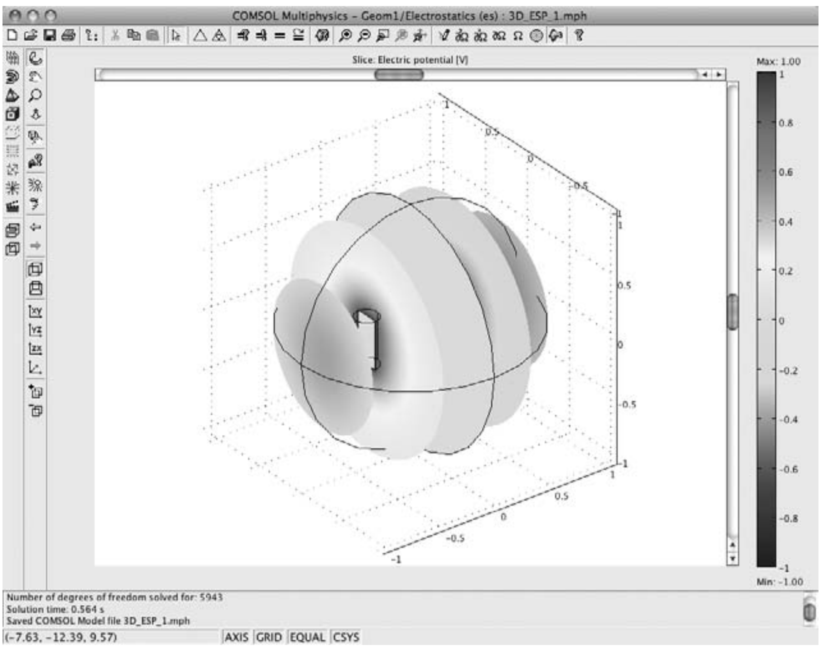


FIGURE 8.78 3D_ESP_1 model default solution plot

FIGURE 8.79 3D_ESP_1 model Plot Parameters selection window

FIGURE 8.80 3D_ESP_1 model Suppress Boundaries selection window

FIGURE 8.81 3D_ESP_1 model streamline plot with suppressed boundaries

Using the menu bar, select Postprocessing > Plot Parameters > Streamline. Click the Apply button, and then click OK. See Figure 8.81.

3D Electrostatic Potential Between Two Cylinders: Summary and Conclusions

The 3D electrostatic potential model presented here demonstrates one of the methods that can be used by the modeler to explore electrostatic potentials in different geometric configurations. This technique can be applied to both scientific and engineering applications (e.g., ranging from X-ray tubes and particle accelerators to paint sprayers and dust precipitators).

3D Electrostatic Potential Between Five Cylinders

To start building the 3D_ESP_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select “3D” from the Space dimension pull-down list. Select COMSOL Multiphysics > Electromagnetics > Electrostatics. See Figure 8.82. Click OK.

FIGURE 8.82 3D_ESP_2 Model Navigator setup

Geometry Modeling

Using the menu bar, select Draw > Sphere. In the Sphere edit window, enter the information shown in Table 8.12. Click OK after filling in the parameters of each separate solid in the appropriate edit window. See Figures 8.83–8.88.

Table 8.12 Geometry Components

Solid	Name	Style	Radius	Height	Axis Base Point (X, Y, Z)	Axis Direction Vector (X, Y, Z)	Figure Number
Sphere	SPH1	Solid	4		(0, 0, 0)	(0, 0, 1)	8.83
Cylinder	CYL1	Solid	0.1	0.4	(−0.4, 0, −0.2)	(0, 0, 1)	8.84
Cylinder	CYL2	Solid	0.1	0.4	(0, 0, −0.2)	(0, 0, 1)	8.85
Cylinder	CYL3	Solid	0.1	0.4	(0.4, 0, −0.2)	(0, 0, 1)	8.86
Cylinder	CYL4	Solid	0.1	0.4	(0, −0.4, −0.2)	(0, 0, 1)	8.87
Cylinder	CYL5	Solid	0.1	0.4	(0, 0.4, −0.2)	(0, 0, 1)	8.88

FIGURE 8.83 3D_ESP_2 model Sphere SPH1 edit window

FIGURE 8.84 3D_ESP_2 model Cylinder CYL1 edit window

FIGURE 8.85 3D_ESP_2 model Cylinder CYL2 edit window

FIGURE 8.86 3D_ESP_2 model Cylinder CYL3 edit window

FIGURE 8.87 3D_ESP_2 model Cylinder CYL4 edit window

FIGURE 8.88 3D_ESP_2 model Cylinder CYL5 edit window

FIGURE 8.89 3D_ESP_2 model Save As edit window

Click the Zoom Extents button. Select File > Save As. Enter 3D_ESP_2.mph in the Save As edit window. See Figure 8.89. Click the Save button.

Using the menu bar, select Draw > Create Composite Object. Enter SPH1-CYL1-CYL2-CYL3-CYL4-CYL5 in the Set formula edit window. See Figure 8.90. Click OK.

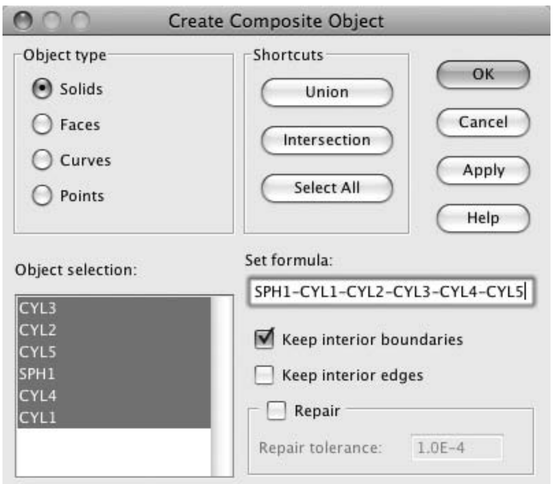


FIGURE 8.90 3D_ESP_2 model Create Composite Object edit window

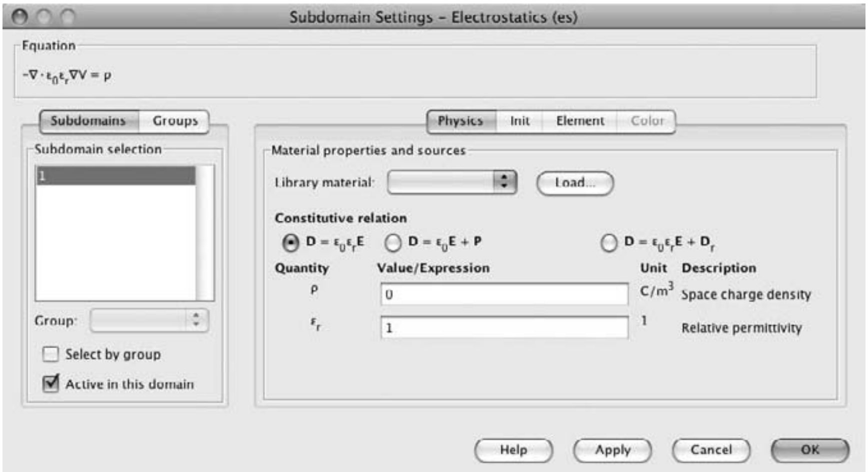


FIGURE 8.91 3D_ESP_2 model Subdomain Settings edit window

Physics Subdomain Settings: Electrostatics (es)

Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 (the only subdomain available). Click the radio button $\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E}$. Enter 0 in the Space charge density (ρ) edit window. Enter 1 in the Relative permittivity (ε_r) edit window. See Figure 8.91. Click OK.

NOTE The numerical value of the permittivity for free space (ε_0) in SI units is $8.854*10^{12}$ F/m. That value is the default value for permittivity incorporated into the COMSOL Multiphysics software. The permittivity of a material is the product of ε_0 and ε_r, which in this case is 1.

Physics Boundary Settings: Electrostatics (es)

Using the menu bar, select Physics > Boundary Settings. Select or enter the settings as indicated in Table 8.13. Click OK. See Figures 8.92, 8.93, and 8.94.

Table 8.13 Boundary Settings

Boundary	Settings	Value	Figure Number
1–4, 23, 24, 28, 29	Zero charge/symmetry	—	8.92
5–14, 19–22, 25, 26, 31–38	Electric potential	1V	8.93
15–18, 27, 30	Electric potential	−1V	8.94

FIGURE 8.92 3D_ESP_2 model Boundary Settings (1–4, 23, 24, 28, 29) edit window

Mesh Generation

Using the menu bar, select Mesh > Free Mesh Parameters. Select “Coarser” from the Predefined mesh sizes pull-down list. See Figure 8.95. Click OK.

Using the menu bar, select Mesh > Initialize Mesh. See Figure 8.96.

FIGURE 8.93 3D_ESP_2 model Boundary Settings (5–14, 19–22, 25, 26, 31–38) edit window

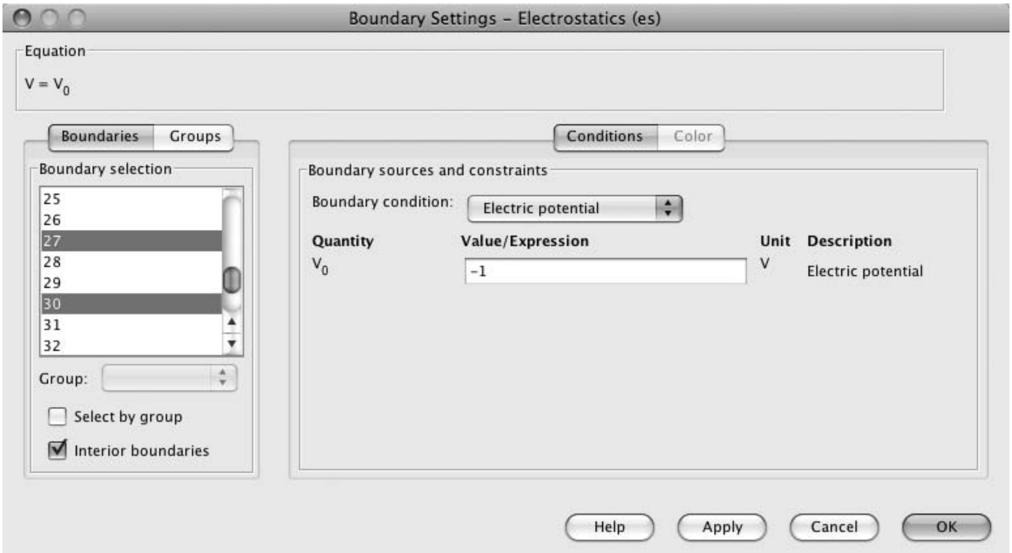


FIGURE 8.94 3D_ESP_2 model Boundary Settings (15–18, 27, 30) edit window

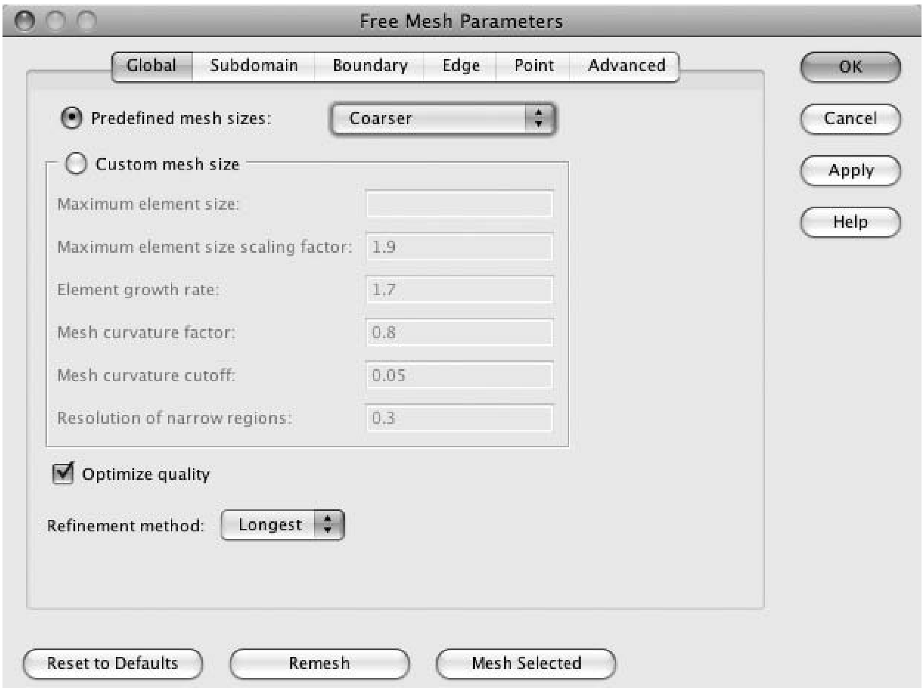


FIGURE 8.95 3D_ESP_2 model Free Mesh Parameters edit window

FIGURE 8.96 3D_ESP_2 model mesh

Solving the 3D_ESP_2 Model

NOTE Electrostatics problems can be complex and difficult. The modeler can, of course, accept the COMSOL software default settings. However, in this case, depending on the modeler's platform, it will probably be best to choose an iterative solver (GMRES) and an appropriate preconditioner (Algebraic multigrid). These choices will reduce both the memory required and the time to solution.

Select Solve > Solver Parameters. Select "GMRES" from the Linear system solver pull-down list. Select "Algebraic multigrid" from the Preconditioner pull-down list. See Figure 8.97.

Click OK. Select Solve > Solve Problem.

Postprocessing and Visualization

The default solution plot is the slice plot. See Figure 8.98.

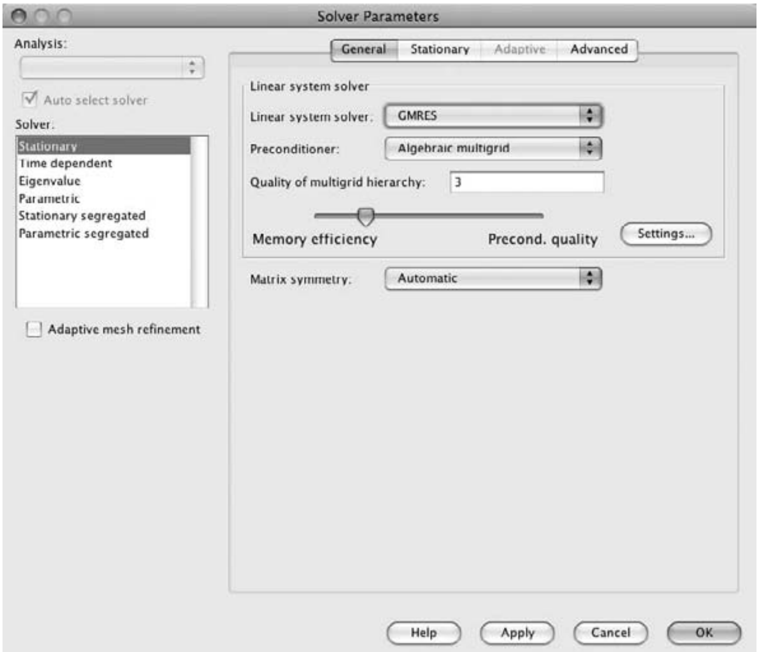


FIGURE 8.97 3D_ESP_2 model Solver Parameters edit window

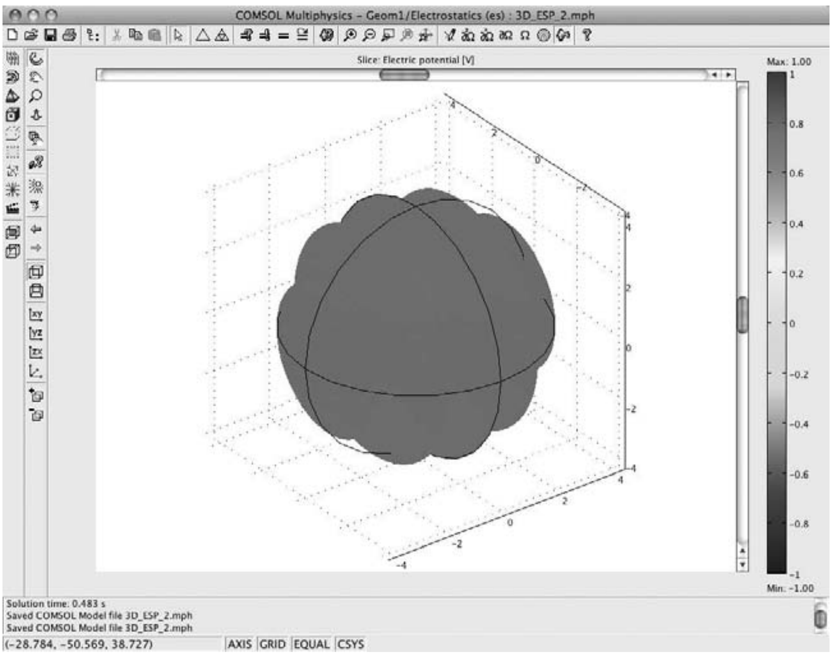


FIGURE 8.98 3D_ESP_2 model default solution plot

FIGURE 8.99 3D_ESP_2 model Plot Parameters selection window

Using the menu bar, select Postprocessing > Plot Parameters > General. Uncheck the Slice check box, and check the Boundary and Streamline check boxes. See Figure 8.99.

Click the Boundary tab. Click the Apply button.

Click the Streamline tab. Enter 29 in the Number of start points window. Click OK.

To see the streamline plot, the modeler will need to suppress the boundaries of the sphere. Using the menu bar, select Options > Suppress > Suppress Boundaries. Select boundaries 1–4, 23, 24, 28, and 29. See Figure 8.100. Click OK.

Using the menu bar, select Postprocessing > Plot Parameters > Streamline. Click the Apply button, and then click OK. See Figure 8.101.

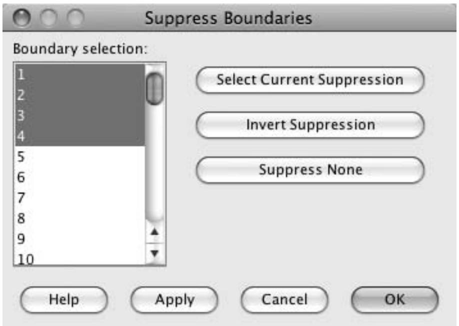


FIGURE 8.100 3D_ESP_2 model Suppress Boundaries selection window

3D Electrostatic Potential Between Five Cylinders: Summary and Conclusions

The 3D electrostatic potential models presented here demonstrate one of the methods that can be used by the new modeler to explore electrostatic potentials in different geometric configurations. The 3D_ESP_2 model is typical of those that might be found in a particle beam analyzer or similar engineering or scientific device. This modeling technique can be applied widely to both scientific and engineering applications (e.g., ranging from X-ray tubes and particle accelerators to paint sprayers and dust precipitators).

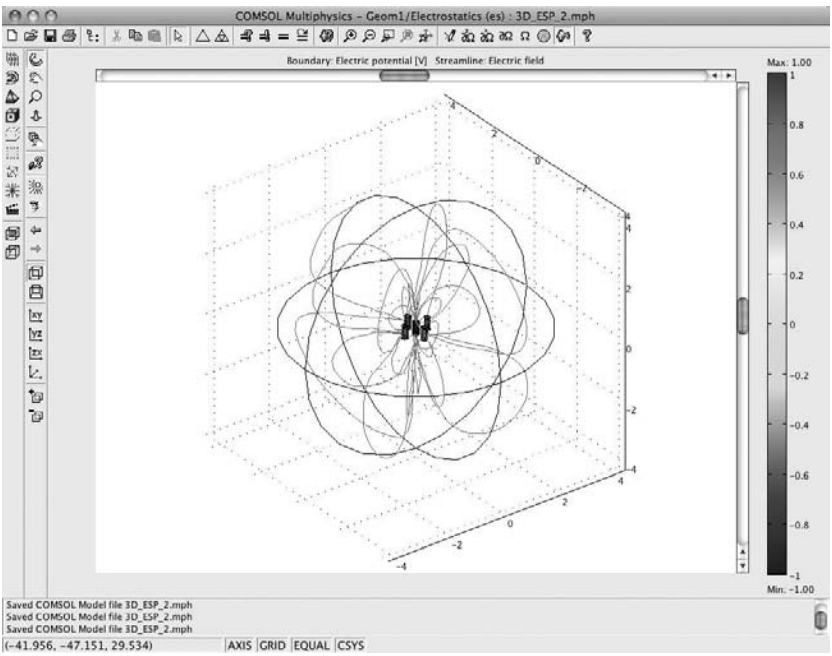


FIGURE 8.101 3D_ESP_2 model streamline plot with suppressed boundaries

Magnetostatic Modeling Basics

The fundamental equations governing electromagnetic phenomena are Maxwell's equations,[17] first published in 1873 by James Clerk Maxwell.[18] Maxwell's equations, as written for free charge and as commonly seen in scientific papers and textbooks, in SI units are

$$\nabla \cdot \mathbf{D} = \rho \tag{8.17}$$

$$\nabla \cdot \mathbf{B} = 0 \tag{8.18}$$

$$\nabla \times \mathbf{E} = -\frac{\partial B}{\partial t} \tag{8.19}$$

$$\nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial D}{\partial t} \tag{8.20}$$

where $\mathbf{E}$ = electric field vector in volts per meter (V/m)
$\mathbf{D}$ = electric flux density in coulombs per square meter (C/m^2)
$\mathbf{B}$ = magnetic field vector in tesla (T)
$\mathbf{H}$ = magnetizing field vector in amperes per meter (A/m)
$\mathbf{J}$ = free current density in amperes per square meter (A/m^2)
ρ = free charge density in coulombs per cubic meter (C/m^3)

To solve any of the potential electromagnetic problems, the modeler also needs to assume continuity (no sinks or sources—that is, "What goes in, comes out"). The equation of continuity is

$$\nabla \cdot \mathbf{J} = -\frac{\partial \rho}{\partial t} \tag{8.21}$$

The modeler also needs to define the properties of the medium(s) throughout which the electromagnetic wave is traveling. These equations are called the constitutive relationships for the medium:

$$\mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P} \tag{8.22}$$

$$\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M}) \tag{8.23}$$

$$\mathbf{J} = \sigma \mathbf{E} \tag{8.24}$$

where $\mathbf{E}$ = electric field vector in volts per meter (V/m)
$\mathbf{D}$ = electric flux density vector in coulombs per square meter (C/m^2)

$\mathbf{P}$ = electric polarization vector in coulombs per square meter (C/m^2)
$\mathbf{B}$ = magnetic field vector in tesla (T)
$\mathbf{H}$ = magnetizing field vector in amperes per meter (A/m)
$\mathbf{M}$ = magnetization vector in amperes per meter (A/m)
$\mathbf{J}$ = free current density in amperes per square meter (A/m^2)
ε_0 = permittivity of vacuum in farads per meter (F/m)
μ_0 = permeability of vacuum in henries per meter (H/m)
σ = electric conductivity in siemens per meter (S/m)

In a magnetostatic model, such as the Helmholtz coil, all parameters are stable and do not fluctuate. If they do fluctuate, it is at a slow rate.

NOTE If the parameters of the model do fluctuate, a good measure of the validity of the model is that the dimensions of the model should be at least 10 times smaller than the wavelength of the fluctuation. Consider, for example, 60 Hz. The wavelength is calculated as follows:

$$\lambda = \frac{c}{f} = \frac{3 \times 10^8}{60} = 5 \times 10^6 \text{ m} \tag{8.25}$$

where λ = wavelength in meters (m)
c = speed of light in meters per second (m/s)
f = frequency in cycles per second (cycle/s)

In the case of a magnetostatic model, the relationships between the potentials and the fields are as follows:

$$\nabla \times (\mu^{-1} \nabla \times \mathbf{A}) = \mathbf{J^e} \tag{8.26}$$

$$\mathbf{B} = \nabla \times \mathbf{A} \tag{8.27}$$

$$\mathbf{H} = \mu^{-1}\mathbf{B} \tag{8.28}$$

$$\mu = \mu_0 \mu_r \tag{8.29}$$

where $\mathbf{A}$ = magnetic vector potential in volt-seconds per meter (V·s/m)
$\mathbf{B}$ = magnetic field vector in tesla (T)
$\mathbf{H}$ = magnetizing field vector in amperes per meter (A/m)
$\mathbf{J^e}$ = externally applied current density in amperes per square meter (A/m^2)
μ_0 = permeability of vacuum in henries per meter (H/m)
μ_r = relative permeability

NOTE The numerical value of the permeability for free space (μ_0) in SI units is exactly $4\pi \times 10^{-7}$ H/m. That value is the default value for permeability incorporated into the COMSOL Multiphysics software. The permeability of a material is the product of μ_0 and μ_r, which in this case is 1.

Because the electromagnetic potentials do not uniquely define a solution, within a gauge transformation[19] it is necessary to choose a gauge (transformation).[20] This technique is called gauge fixing. In this case, the gauge chosen is called the Coulomb gauge. The condition of the Coulomb gauge is that $\nabla \cdot \mathbf{A} = 0$.

To avoid numerical instability in the model, $\nabla \cdot \mathbf{A}$ is numerically adjusted to zero by using a type of special pre- and post-smoother called an SOR gauge.[21]

3D Magnetic Field of a Helmholtz Coil

The following numerical solution model (3D_HC_1 model) is derived from a model that was originally developed by COMSOL as an AC/DC Module Electrical Components demonstration model. That model was developed for distribution with the Multiphysics software as a COMSOL AC/DC Module Model Library.

To start building the 3D_HC_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select AC/DC Module > Statics > Magnetostatics. See Figure 8.102. Click OK.

FIGURE 8.102 3D_HC_1 Model Navigator setup

Table 8.14 Constants Edit Window

Name	Expression	Description	Figure Number
J0	1[A/m^2]	Coil current density	8.103

Constants

Using the menu bar, select Options > Constants. Enter the constant shown in Table 8.14. Click OK. See Figure 8.103.

Select File > Save As. Enter 3D_HC_1.mph in the Save As edit window. See Figure 8.104. Click the Save button.

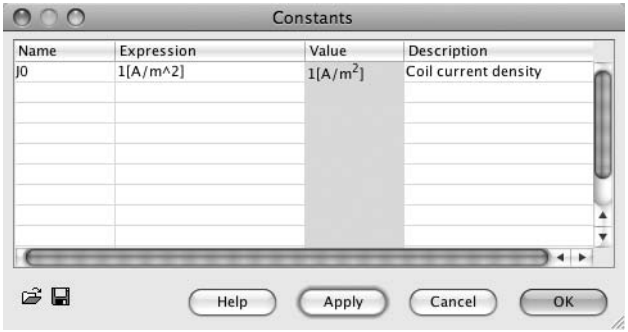


FIGURE 8.103 3D_HC_1 model Constants edit window

FIGURE 8.104 3D_HC_1 model Save As edit window

FIGURE 8.105 3D_HC_1 model Work-Plane Settings edit window

Geometry Modeling

Each Helmholtz coil is created in cross section by drawing squares in the 2D work-plane. The modeler then creates a solid coil by revolution (revolve) of the 2D work-plane geometry into the 3D geometry.

Using the menu bar, select Draw > Work-Plane Settings. See Figure 8.105. Click OK.

Using the menu bar, select Draw > Specify Objects > Square. In the Square edit window, enter the information shown in Table 8.15. Click OK after filling in the parameters of each separate square in the appropriate edit window. See Figures 8.106 and 8.107.

Table 8.15 Geometry Components

Name	Width	Base	x	y	Figure Number
SQ1	0.05	Corner	−0.425	0.175	8.106
SQ2	0.05	Corner	−0.425	−0.225	8.107

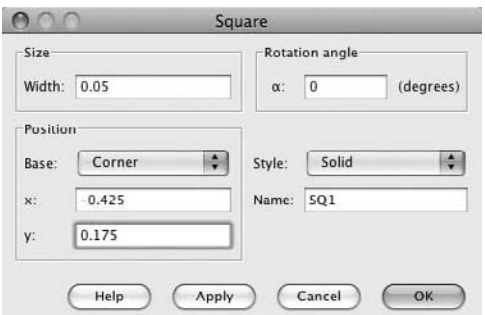


FIGURE 8.106 3D_HC_1 model Square SQ1 edit window

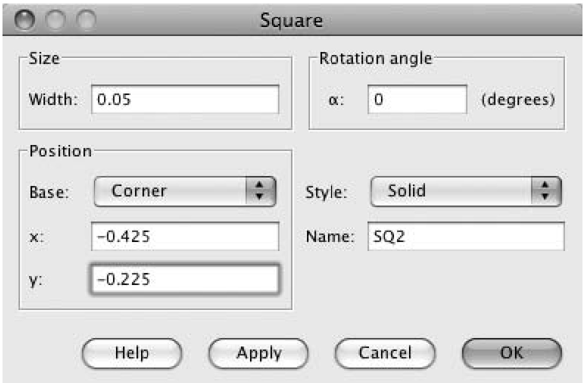


FIGURE 8.107 3D_HC_1 model Square SQ2 edit window

Click the Zoom Extents button. Select SQ1 and SQ2. Using the menu bar, select Draw > Revolve. See Figure 8.108.

Click OK. See Figure 8.109.

Using the menu bar, select Draw > Sphere. Enter 1 in the Radius edit window. See Figure 8.110.

Click OK, and then click the Zoom Extents button. Click on the display window background outside the sphere. See Figure 8.111.

FIGURE 8.108 3D_HC_1 model Revolve edit window

FIGURE 8.109 3D_HC_1 model Helmholtz coil pair

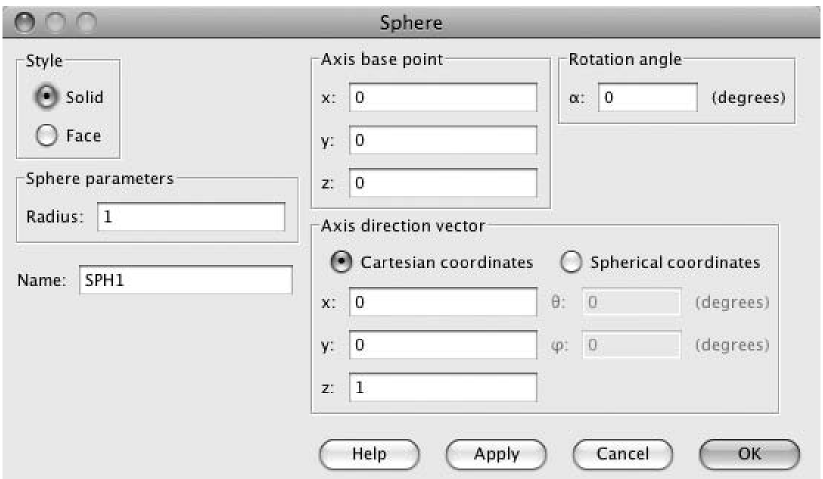


FIGURE 8.110 3D_HC_1 model Sphere edit window

FIGURE 8.111 3D_HC_1 model sphere and Helmholtz coil pair

Physics Subdomain Settings: Magnetostatics (emqa)

Using the menu bar, select Physics > Subdomain Settings. Select the subdomain(s) and enter the expression indicated in Table 8.16. Click OK. See Figures 8.112 and 8.113.

NOTE The numerical value of the permeability for free space (μ_0) in SI units is exactly $4\pi \times 10^{-7}$ H/m. That value is the default value for permeability incorporated into the COMSOL Multiphysics software. The permeability of a material is the product of μ_0 and μ_r, which in this case is 1.

Table 8.16 Subdomain Settings

Subdomain	Quantity	Value/Expression	Figure Number
1	$\mathbf{J}^e$	0 0 0	8.112
2, 3	$\mathbf{J}^e$	−J0*z/sqrt(x^2+z^2) 0 J0*x/sqrt(x^2+z^2)	8.113

FIGURE 8.112 3D_HC_1 model Subdomain Settings (1) edit window

Physics Boundary Settings: Magnetostatics (emqa)

Using the menu bar, select Physics > Boundary Settings. Verify or enter the default boundary setting (Magnetic insulation) for spherical boundaries 1–4, 21, 22, 31, and 32. See Figure 8.114. Click OK.

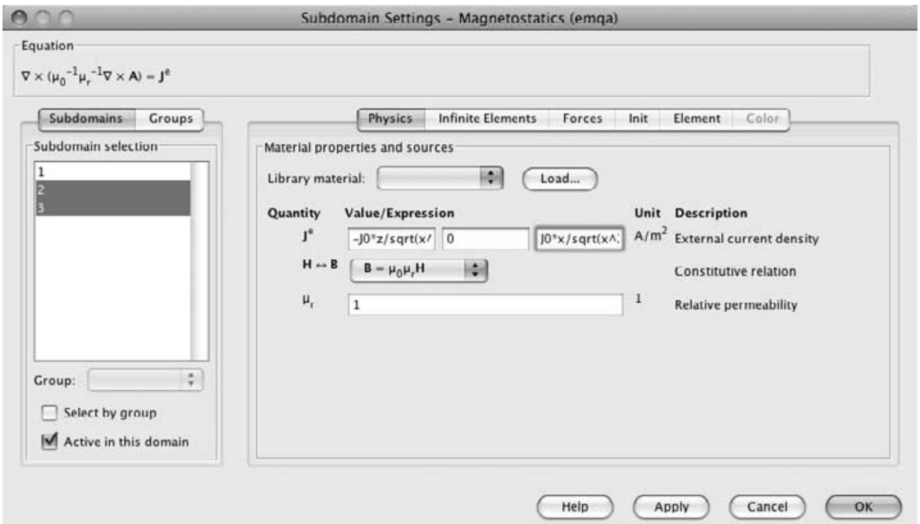


FIGURE 8.113 3D_HC_1 model Subdomain Settings (2, 3) edit window

FIGURE 8.114 3D_HC_1 model Boundary Settings (1–4, 21, 22, 31, 32) edit window

Mesh Generation

Using the menu bar, select Mesh > Free Mesh Parameters. Select "Coarser" from the Predefined mesh sizes pull-down list. See Figure 8.115.

Click the Subdomain tab. Select subdomains 2 and 3. Enter 0.05 in the Maximum element size edit window. See Figure 8.116.

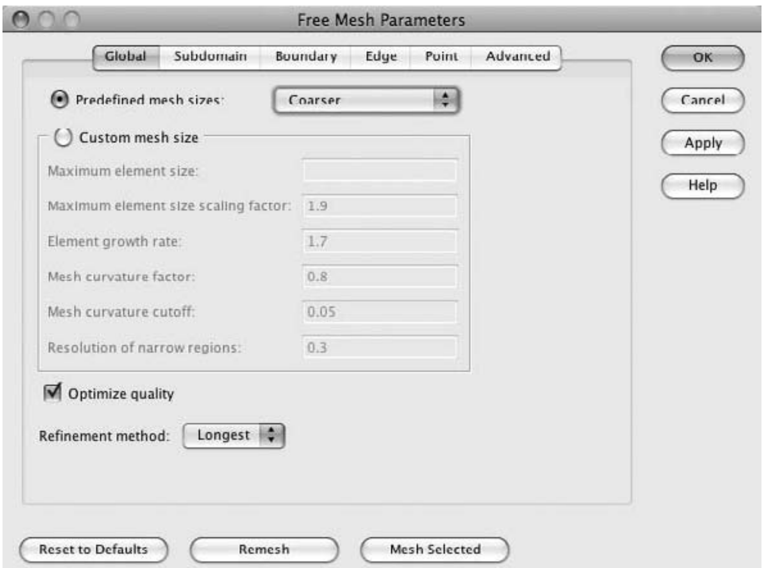


FIGURE 8.115 3D_HC_1 model Free Mesh Parameters edit window, Global tab

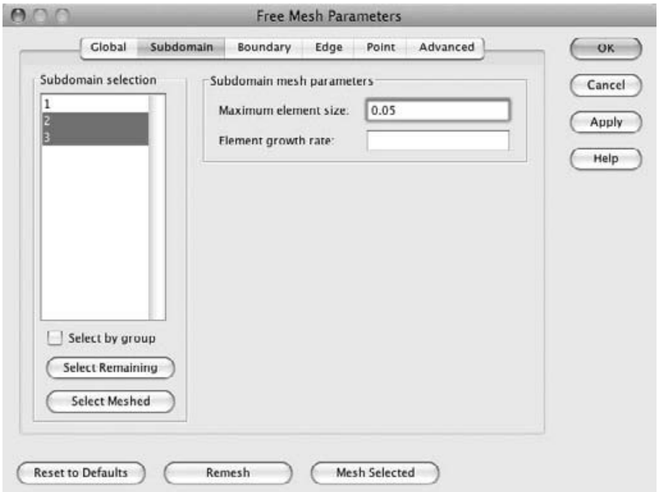


FIGURE 8.116 3D_HC_1 model Free Mesh Parameters edit window, Subdomain tab

Click the Remesh button, and then click OK. See Figure 8.117.

Solving the 3D_HC_1 Model

Using the menu bar, select Solve > Solve Problem.

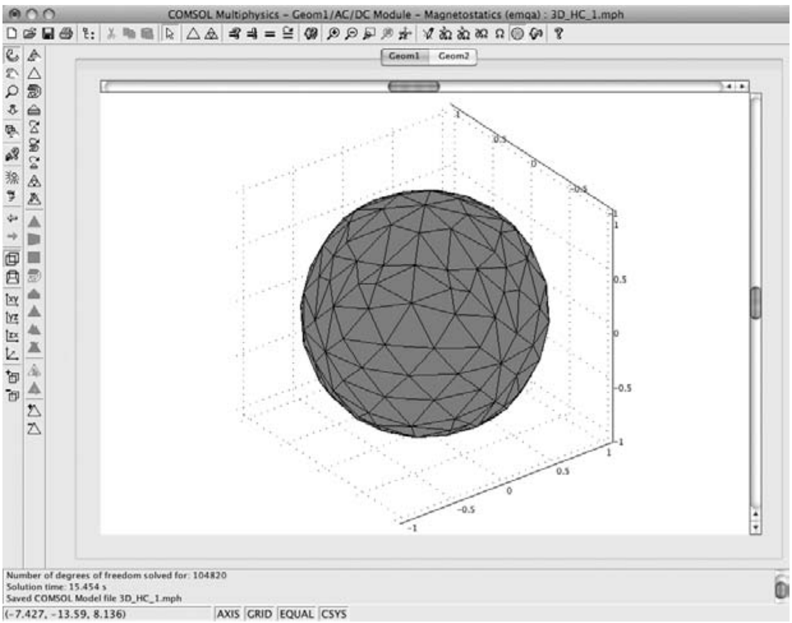


FIGURE 8.117 3D_HC_1 model mesh

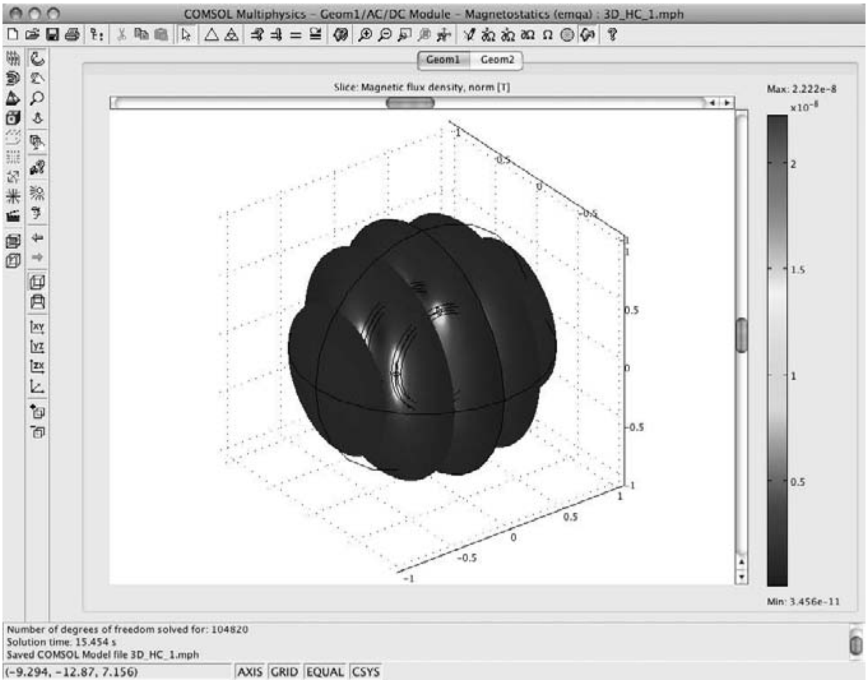


FIGURE 8.118 3D_HC_1 model default solution plot

Postprocessing and Visualization

The default solution plot is the slice plot. See Figure 8.118.

To see the model solution inside the sphere, the modeler needs to suppress the boundaries of the sphere. Using the menu bar, select Options > Suppress > Suppress Boundaries. Select boundaries 1–4, 21, 22, 31, and 32 in the Boundary selection window. See Figure 8.119. Click OK.

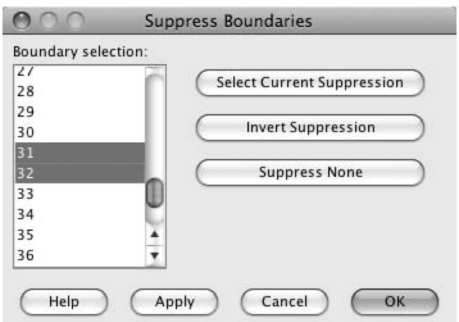


FIGURE 8.119 3D_HC_1 model plot Suppress Boundaries selection window

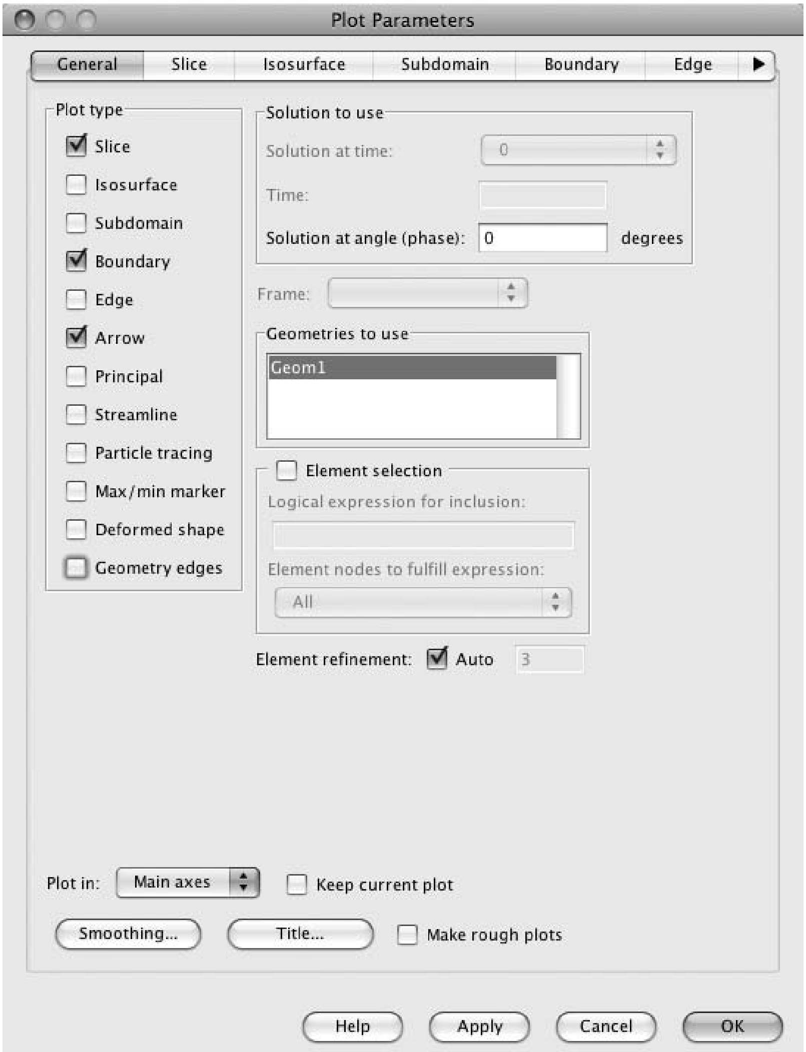


FIGURE 8.120 3D_HC_1 model Plot Parameters selection window, General tab

Using the menu bar, select Postprocessing > Plot Parameters > General. Check the Slice, Boundary, and Arrow check boxes. Uncheck the Geometry edges check box. See Figure 8.120. Click the Apply button.

Click the Slice tab. Select or verify the Predefined quantities: Magnetic flux density, norm. Enter 0 in the x and y levels edit windows and 1 in the z levels edit window. See Figure 8.121. Click the Apply button.

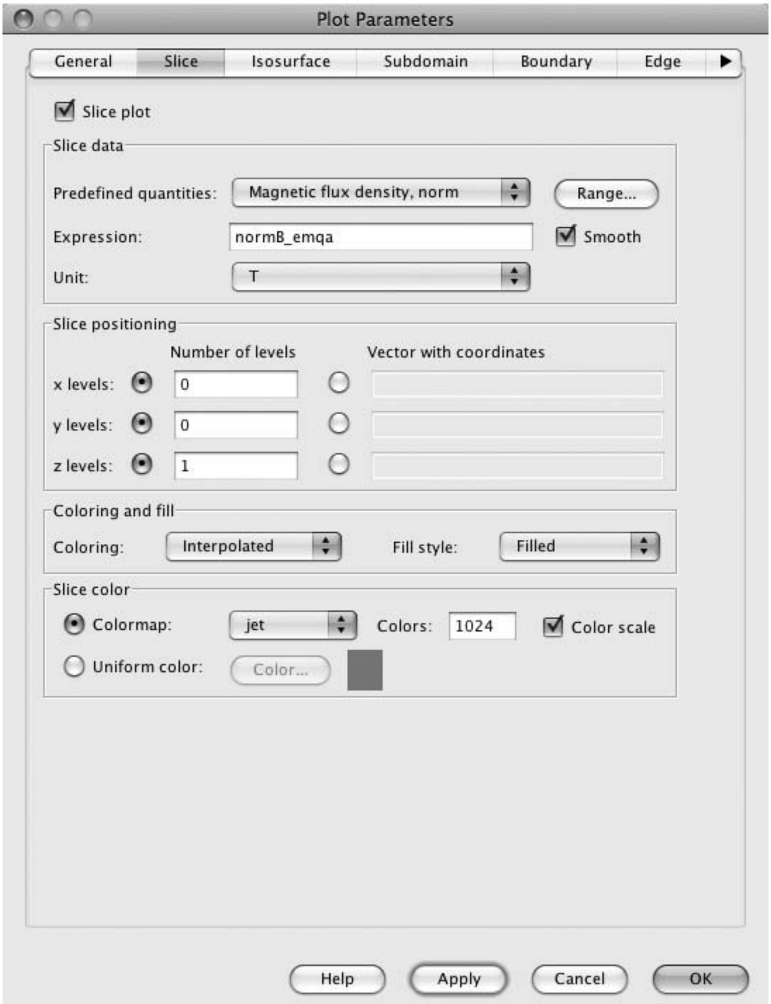


FIGURE 8.121 3D_HC_1 model Plot Parameters selection window, Slice tab

Click the Boundary tab. Enter 1 in the Boundary data Expression edit window. Click the Uniform color radio button and choose black. Click OK. See Figure 8.122. Click the Apply button.

Click the Arrow tab. Select "Magnetic field" from the Predefined quantities pull-down list. Enter 24 in the x points window, 10 in the y points window, and 1 in the z points window. Uncheck the Scale factor Auto check box. Enter 0.5 in the Scale factor edit window. See Figure 8.123. Click the Apply button.

FIGURE 8.122 3D_HC_1 model Plot Parameters selection window, Boundary tab

Click OK. See Figure 8.124.

Cross-Section Field Analysis

To obtain a graphical plot of the magnetic field, use the Cross-Section Plot feature. Using the menu bar, select Postprocessing > Cross-Section Plot Parameters > General. Click the Line/Extrusion plot radio button in the Plot type selection window. Click the Line/Extrusion tab. Select “Magnetic flux density, norm” from the Predefined quantities pull-down list in the y-axis data selection window. Select “x” from the x-axis data

FIGURE 8.123 3D_HC_1 model Plot Parameters selection window, Arrow tab

pull-down list. In the Cross-section line data edit windows, enter −0.8 for x0 and 0.8 for x1. See Figure 8.125.

Click the Apply button. See Figure 8.126.

Click OK. See Figure 8.127.

3D Magnetic Field of a Helmholtz Coil: Summary and Conclusions

The 3D magnetic field of a Helmholtz coil model demonstrates the magnetic field uniformity of a Helmholtz coil pair. The magnetostatic modeling technique can be applied to a diverse collection of scientific and engineering applications (e.g., ranging from magnetometers and Hall effect sensors to biomagnetic and medical studies).

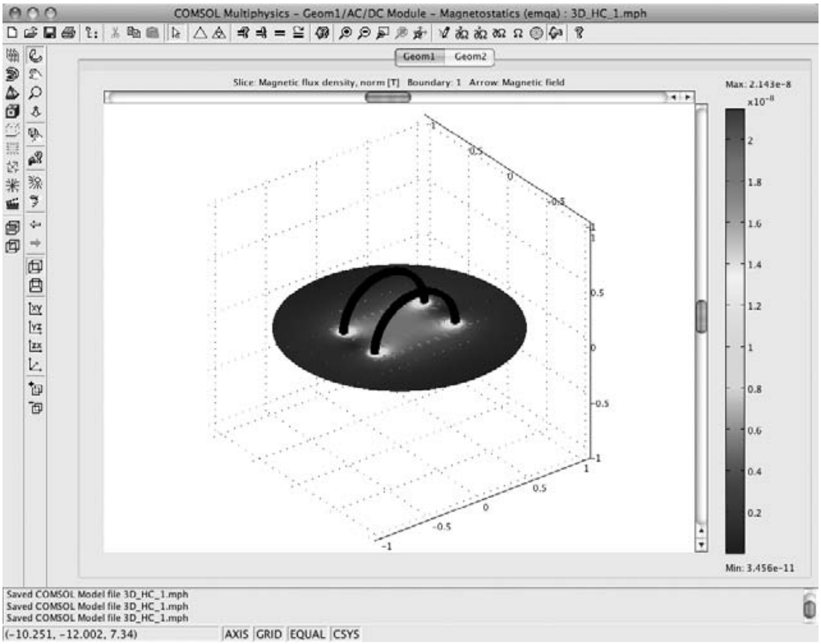


FIGURE 8.124 3D_HC_1 model solution plot

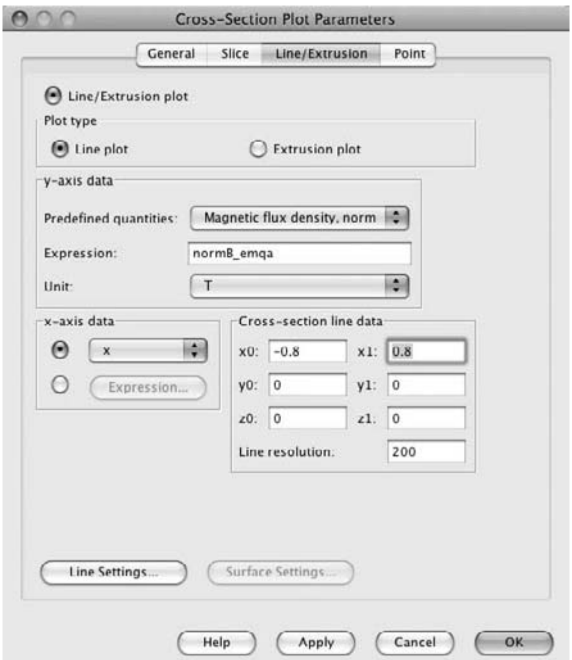


FIGURE 8.125 3D_HC_1 model Cross-Section Plot Parameters edit window

FIGURE 8.126 3D_HC_1 model cross-section plot

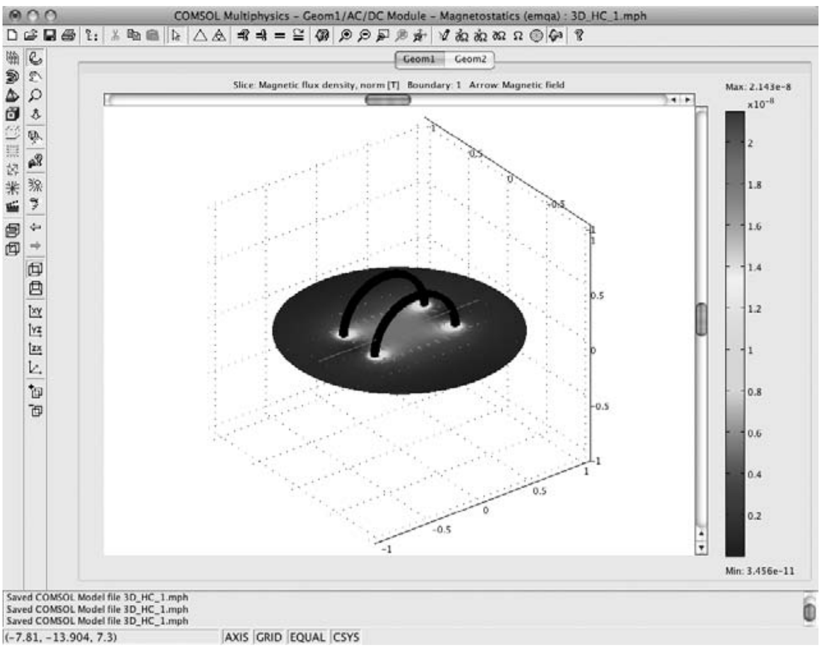


FIGURE 8.127 3D_HC_1 model solution plot with cross-section plot line

FIGURE 8.128 3D_HC_2 Model Navigator setup

3D Magnetic Field of a Helmholtz Coil with a Magnetic Test Object

The following numerical solution model (3D_HC_2 model) is derived from a model that was originally developed by COMSOL as an AC/DC Module Electrical Components demonstration model. That model was developed for distribution with the Multiphysics software as a COMSOL AC/DC Module Model Library.

To start building the 3D_HC_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select AC/DC Module > Statics > Magnetostatics. See Figure 8.128. Click OK.

Constants

Using the menu bar, select Options > Constants. Enter the constant shown in Table 8.17. Click OK. See Figure 8.129.

Table 8.17 Constants Edit Window

Name	Expression	Description	Figure Number
J0	1[A/m^2]	Coil current density	8.129

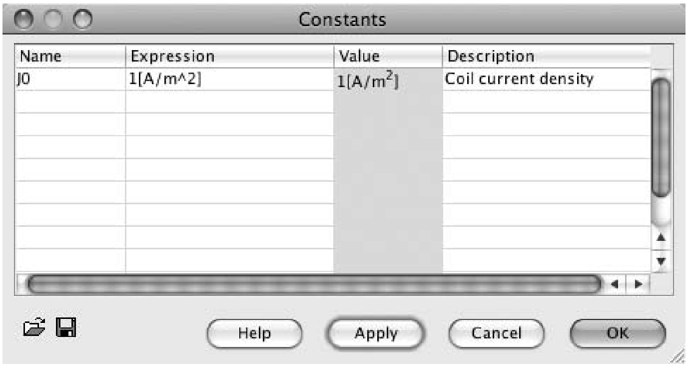


FIGURE 8.129 3D_HC_2 model Constants edit window

Select File > Save As. Enter 3D_HC_2.mph in the Save As edit window. See Figure 8.130. Click the Save button.

Geometry Modeling

Each Helmholtz coil is created in cross section by drawing squares in the 2D work-plane. The modeler then creates a solid coil by revolution (revolve) of the 2D work-plane geometry into the 3D geometry.

FIGURE 8.130 3D_HC_2 model Save As edit window

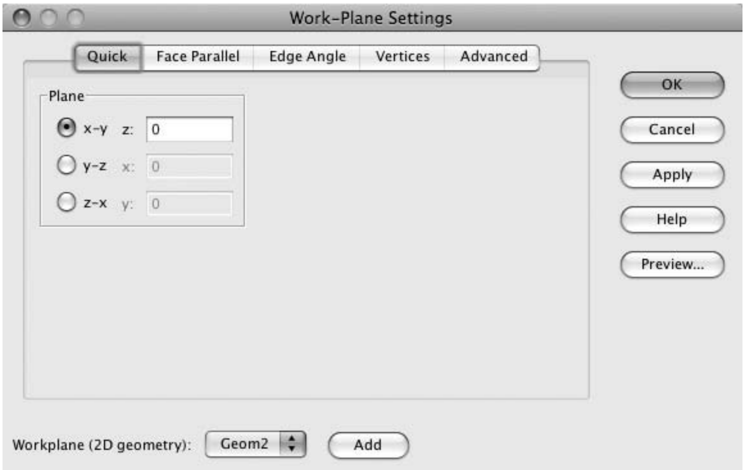


FIGURE 8.131 3D_HC_2 model Work-Plane Settings edit window

Using the menu bar, select Draw > Work-Plane Settings. See Figure 8.131. Click OK.

Using the menu bar, select Draw > Specify Objects > Square. In the Square edit window, enter the information shown in Table 8.18. Click OK after filling in the parameters of each separate square in the appropriate edit window. See Figures 8.132 and 8.133.

Table 8.18 Geometry Components

Name	Width	Base	x	y	Figure Number
SQ1	0.05	Corner	−0.425	0.175	8.132
SQ2	0.05	Corner	−0.425	−0.225	8.133

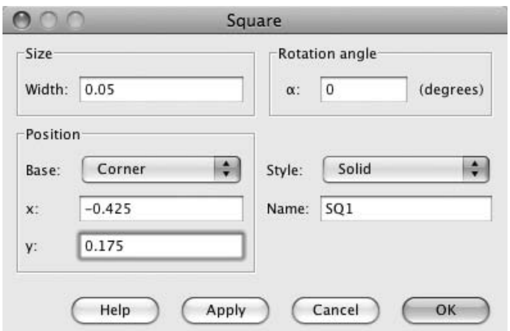


FIGURE 8.132 3D_HC_2 model Square SQ1 edit window

FIGURE 8.133 3D_HC_2 model Square SQ2 edit window

Click the Zoom Extents button. Select SQ1 and SQ2. Using the menu bar, select Draw > Revolve. See Figure 8.134.

Click OK. See Figure 8.135.

Using the menu bar, select Draw > Ellipsoid. Enter x = 0.05, y = 0.3, and z = 0.05 in the Length semiaxes edit windows. Enter x = 0, y = 0 and z = 1 in the Cartesian coordinates edit windows. See Figure 8.136. Click OK.

Using the menu bar, select Draw > Sphere. Enter 1 in the Radius edit window. See Figure 8.137. Click OK.

Click the Zoom Extents button. Click on the display window background outside the sphere. See Figure 8.138.

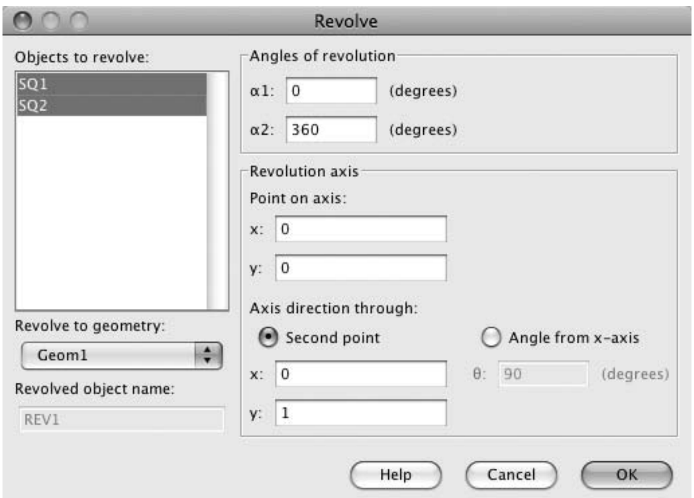


FIGURE 8.134 3D_HC_2 model Revolve edit window

FIGURE 8.135 3D_HC_2 model Helmholtz coil pair

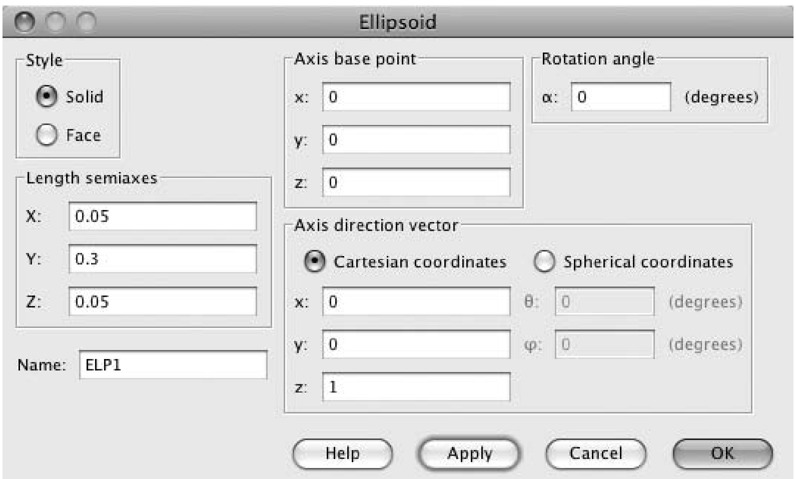


FIGURE 8.136 3D_HC_2 model Ellipsoid edit window

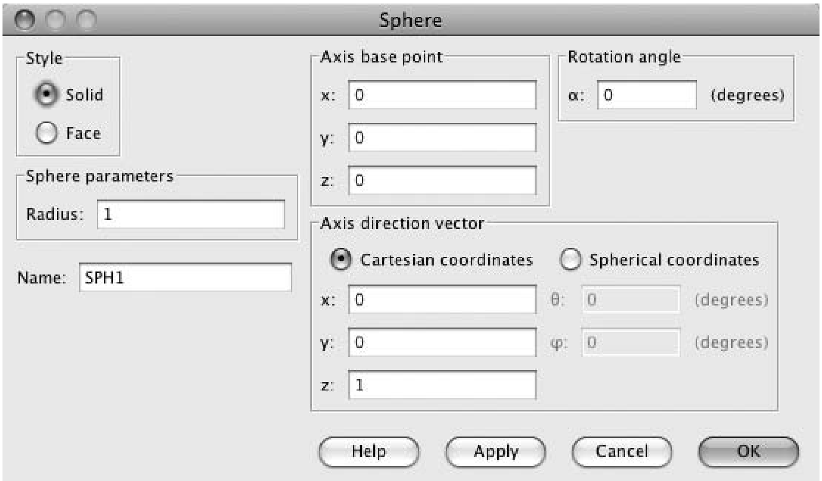


FIGURE 8.137 3D_HC_2 model Sphere edit window

FIGURE 8.138 3D_HC_2 model, sphere, Helmholtz coil pair, and magnetic ellipsoid

Table 8.19 Subdomain Settings

Subdomain	Quantity	Value/Expression	Figure Number
1	$\mathbf{J^e}$	0 0 0	8.139
	μ_r	1	
2, 3	$\mathbf{J^e}$	−J0*z/sqrt(x^2+z^2) 0 J0*x/sqrt(x^2+z^2)	8.140
	μ_r	1	
4	$\mathbf{J^e}$	0 0 0	8.141
	μ_r	15000	

Physics Subdomain Settings: Magnetostatics (emqa)

Using the menu bar, select Physics > Subdomain Settings. Select the Subdomain(s) and enter the expression indicated in Table 8.19. Click OK. See Figures 8.139, 8.140, and 8.141.

NOTE The numerical value of the permeability for free space (μ_0) in SI units is exactly $4\pi \times 10^{-7}$ H/m. That value is the default value for permeability incorporated into the COMSOL Multiphysics software. The permeability of a material is the product of μ_0 and μ_r, which in this case is 1.

FIGURE 8.139 3D_HC_2 model Subdomain Settings (1) edit window

FIGURE 8.140 3D_HC_2 model Subdomain Settings (2, 3) edit window

Physics Boundary Settings: Magnetostatics (emqa)

Using the menu bar, select Physics > Boundary Settings. Verify or enter the default boundary setting (Magnetic insulation) for spherical boundaries 1–4, 25, 26, 37, and 40. See Figure 8.142. Click OK.

FIGURE 8.141 3D_HC_2 model Subdomain Settings (4) edit window

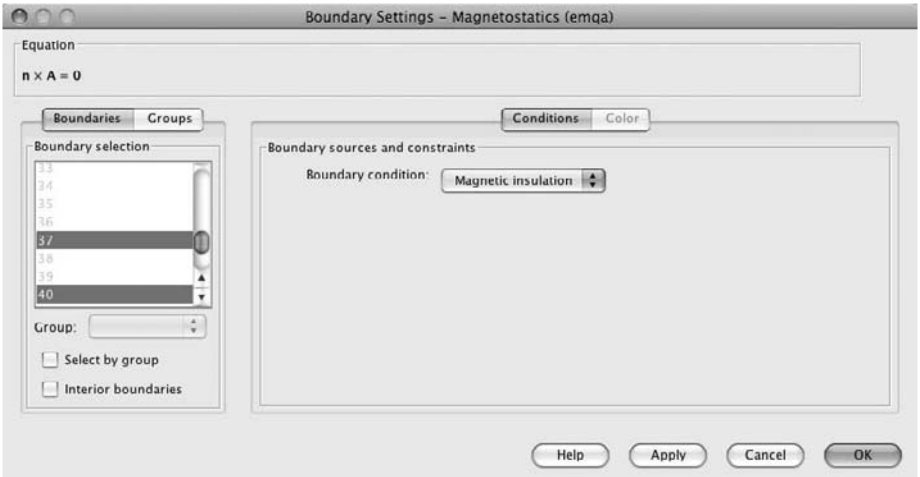


FIGURE 8.142 3D_HC_2 model Boundary Settings (1–4, 25, 26, 37, 40) edit window

Mesh Generation

Using the menu bar, select Mesh > Free Mesh Parameters. Select “Coarser” from the Predefined mesh sizes pull-down list. See Figure 8.143.

Click the Subdomain tab. Select subdomains 2 and 3. Enter 0.05 in the Maximum element size edit window. See Figure 8.144.

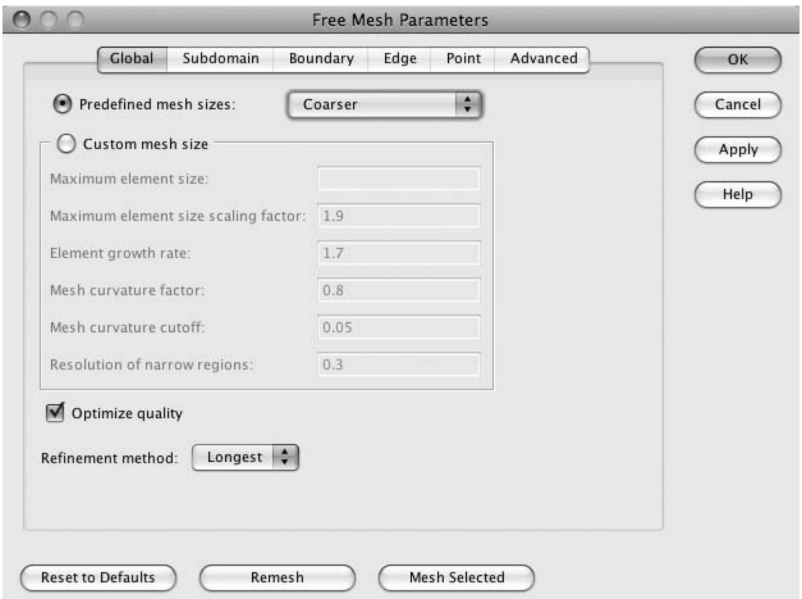


FIGURE 8.143 3D_HC_2 model Free Mesh Parameters edit window, Global tab

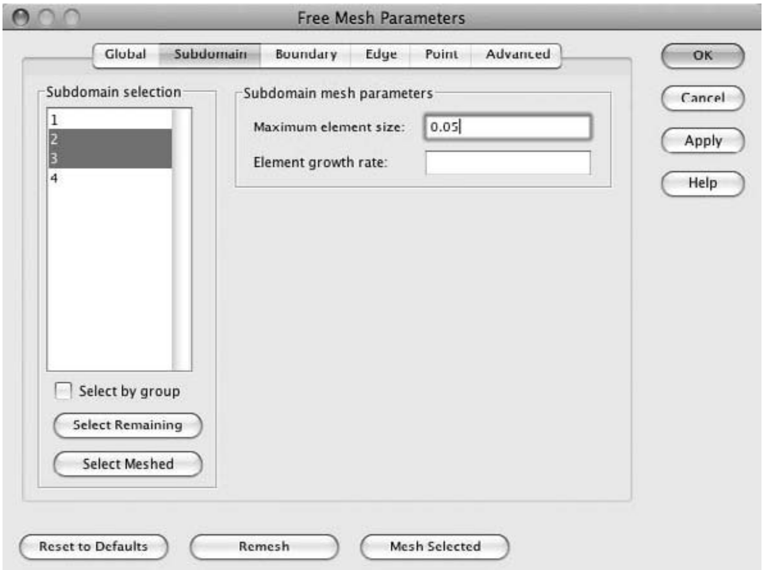


FIGURE 8.144 3D_HC_2 model Free Mesh Parameters edit window, Subdomain (2, 3) tab

Select subdomain 4. Enter 0.03 in the Maximum element size edit window. See Figure 8.145.

Click the Remesh button, and then click OK. See Figure 8.146.

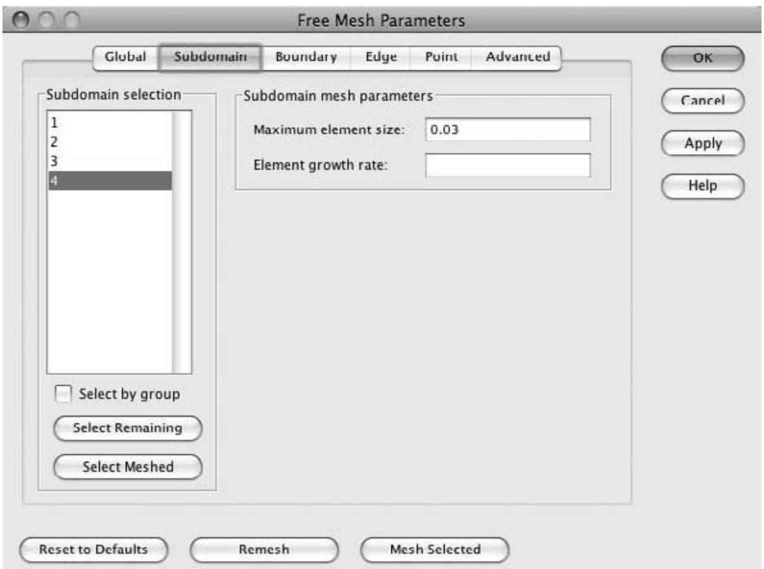


FIGURE 8.145 3D_HC_2 model Free Mesh Parameters edit window, Subdomain (4) tab

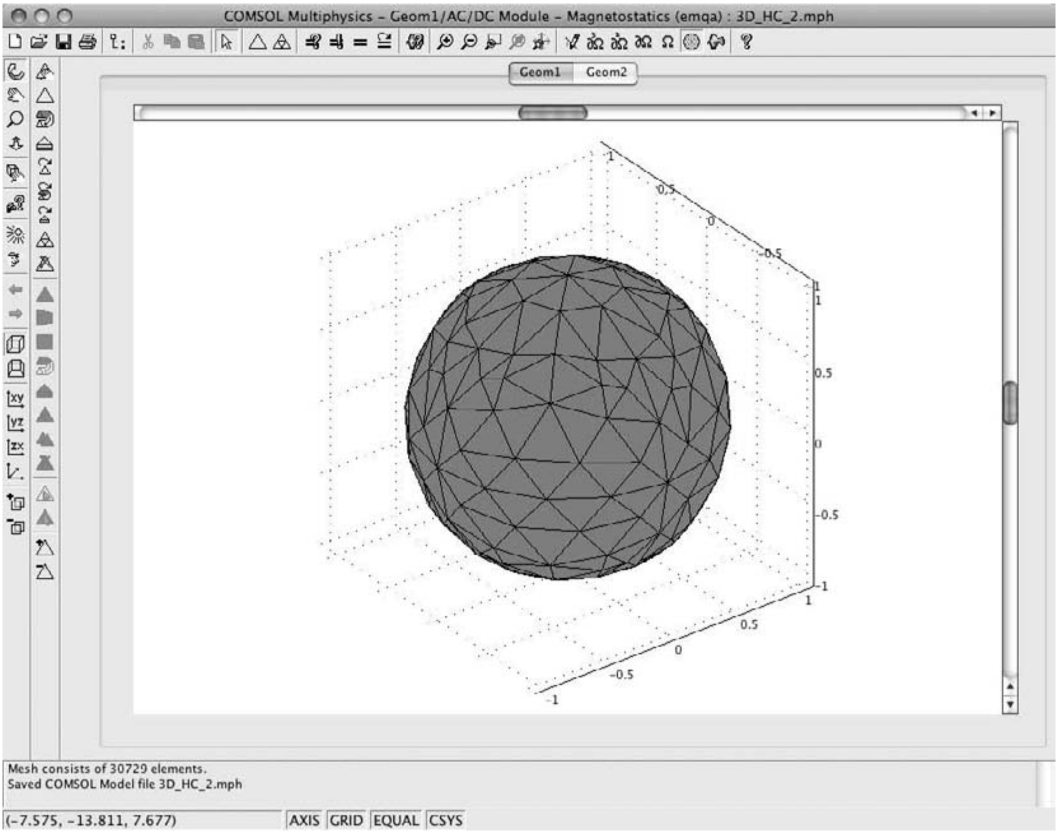


FIGURE 8.146 3D_HC_2 model mesh

Solving the 3D_HC_2 Model

Using the menu bar, select Solve > Solve Problem.

Postprocessing and Visualization

The default solution plot is the slice plot. See Figure 8.147.

To see the model solution inside the sphere, the modeler needs to suppress the boundaries of the sphere. Using the menu bar, select Options > Suppress > Suppress Boundaries. Select boundaries 1–4, 25, 26, 37, and 40 in the Boundary selection window. See Figure 8.148. Click OK.

Using the menu bar, select Postprocessing > Plot Parameters > General. Check the Slice, Boundary, and Arrow check boxes. Uncheck the Geometry edges check box. See Figure 8.149. Click the Apply button.

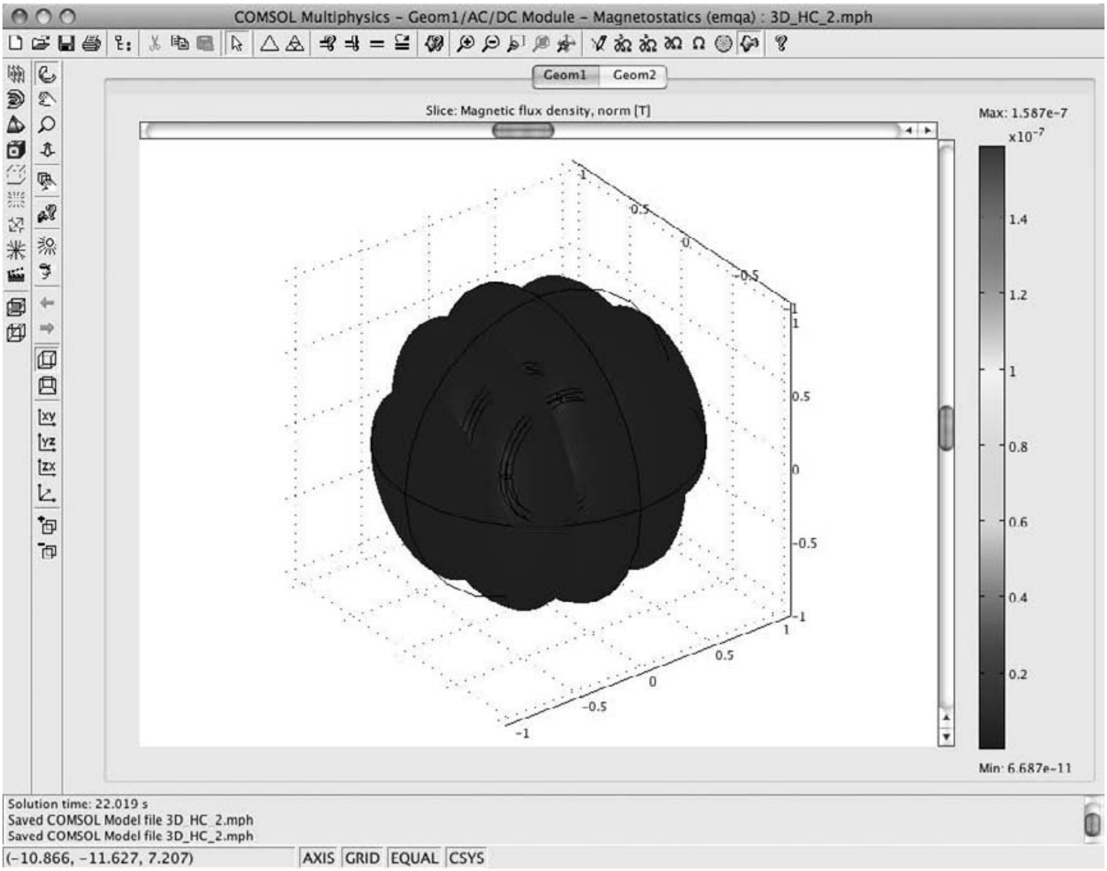


FIGURE 8.147 3D_HC_2 model default solution plot

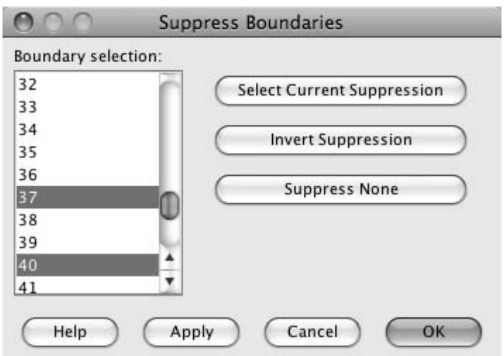


FIGURE 8.148 3D_HC_2 model plot Suppress Boundaries selection window

FIGURE 8.149 3D_HC_2 model Plot Parameters selection window, General tab

Click the Slice tab. Select or verify the Predefined quantities: Magnetic flux density, norm. Enter 0 in the x levels edit window, 0 in the y levels edit window, and 1 in the z levels edit window. See Figure 8.150. Click the Apply button.

Click the Boundary tab. Enter 1 in the Boundary data Expression edit window. Click the Uniform color radio button, and choose black. Click OK. See Figure 8.151. Click the Apply button.

Click the Arrow tab. Select "Magnetic field" from the Predefined quantities pull-down list. Enter 24 in the x points window, 10 in the y points window, and 1 in the z points window. Uncheck the Scale factor Auto check box. Enter 0.5 in the Scale factor edit window. See Figure 8.152. Click the Apply button.

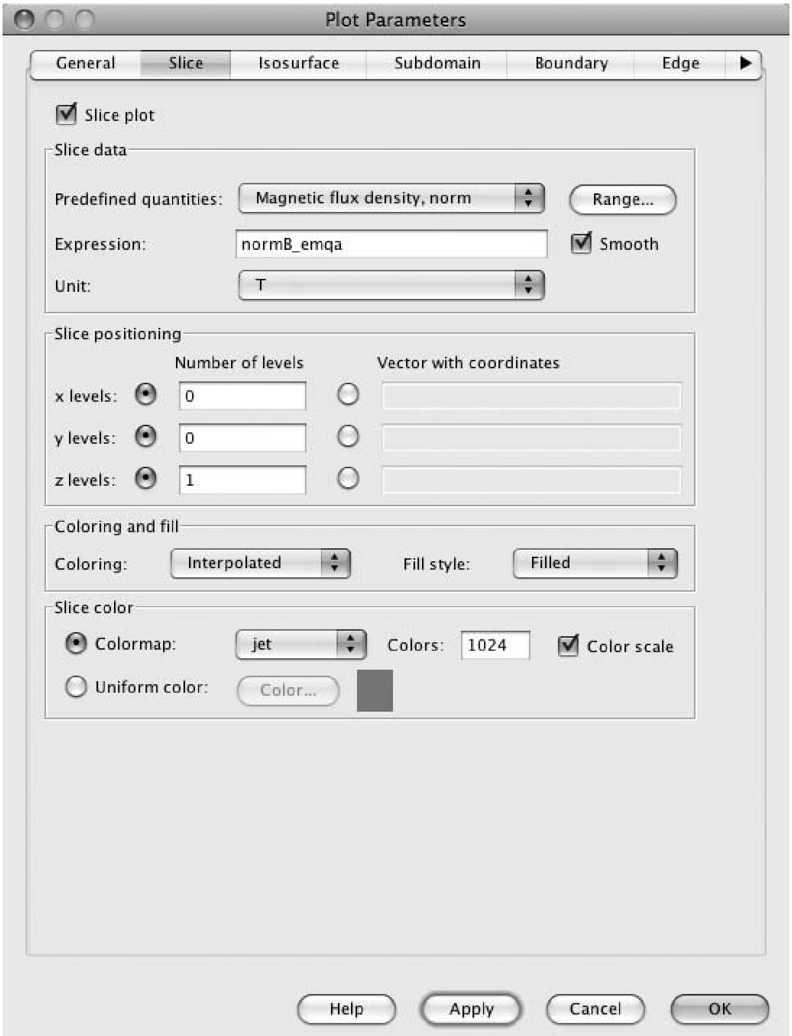


FIGURE 8.150 3D_HC_2 model Plot Parameters selection window, Slice tab

Click OK. See Figure 8.153.

Cross-Section Field Analysis

To obtain a graphical plot of the magnetic field, use the Cross-Section Plot feature. Using the menu bar, select Postprocessing > Cross-Section Plot Parameters > General. Click the Line/Extrusion plot radio button in the Plot type selection window. Click the Line/Extrusion tab. Select “Magnetic flux density, norm” from the Predefined quantities pull-down list in the y-axis data selection window. Select “x” from the x-axis data

FIGURE 8.151 3D_HC_2 model Plot Parameters selection window, Boundary tab

pull-down list. In the Cross-section line data edit windows, enter −0.8 for x0 and 0.8 for x1. See Figure 8.154.

Click the Apply button. See Figure 8.155.

Click OK. See Figure 8.156.

3D Magnetic Field of a Helmholtz Coil with a Magnetic Test Object: Summary and Conclusions

The 3D magnetic field of a Helmholtz coil with a magnetic test object model demonstrates the magnetic field concentration, when a high relative permeability object lies within the field of the Helmholtz coil. This magnetostatic modeling technique can be applied to a diverse collection of scientific and engineering test, measurement, and design applications.

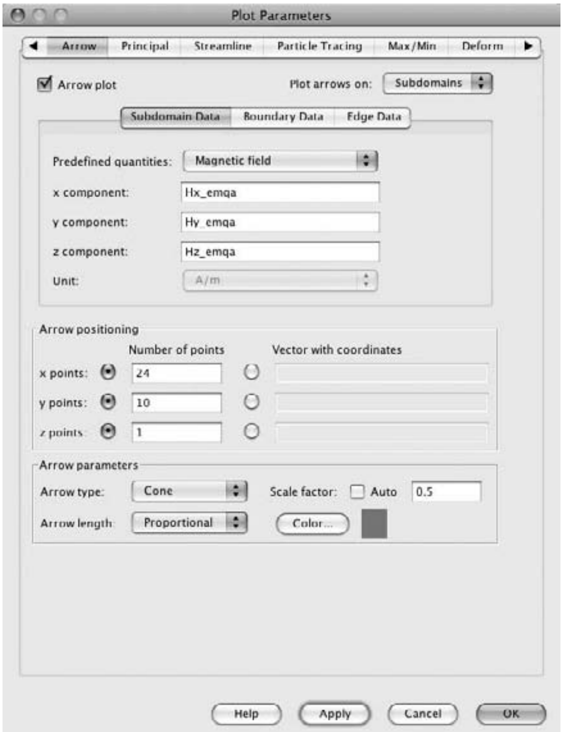


FIGURE 8.152 3D_HC_2 model Plot Parameters selection window, Arrow tab

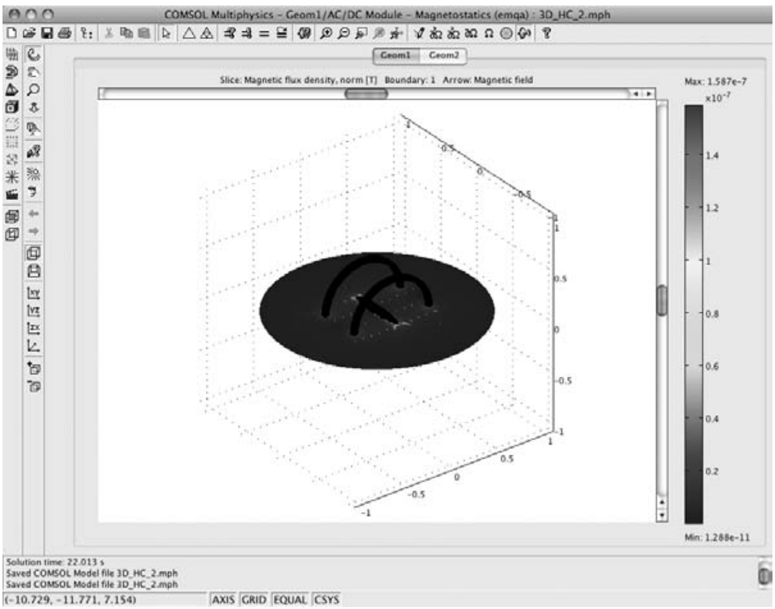


FIGURE 8.153 3D_HC_2 model solution plot

FIGURE 8.154 3D_HC_2 model Cross-Section Plot Parameters edit window

FIGURE 8.155 3D_HC_2 model cross-section plot

FIGURE 8.156 3D_HC_2 model solution plot with cross-section plot line

References

1. http://en.wikipedia.org/wiki/Thin_film
2. http://en.wikipedia.org/wiki/Layer_(electronics)
3. http://en.wikipedia.org/wiki/Coating
4. http://focus.ti.com/lit/an/slyt209a/slyt209a.pdf
5. http://en.wikipedia.org/wiki/Electrical_resistance
6. http://en.wikipedia.org/wiki/Voltage_divider
7. http://en.wikipedia.org/wiki/Thales_of_Miletus
8. http://en.wikipedia.org/wiki/Otto_von_Guericke
9. http://en.wikipedia.org/wiki/Electrostatic_generator
10. http://en.wikipedia.org/wiki/Charles-Augustin_de_Coulomb
11. http://en.wikipedia.org/wiki/Carl_Friedrich_Gauss
12. http://en.wikipedia.org/wiki/Electrostatics

13. http://en.wikipedia.org/wiki/Gauss%27s_Law
14. http://en.wikipedia.org/wiki/Laplacian_operator
15. http://en.wikipedia.org/wiki/Vector_Laplacian
16. http://en.wikipedia.org/wiki/Pierre-Simon_de_Laplace
17. http://en.wikipedia.org/wiki/Maxwell%27s_Equations
18. http://en.wikipedia.org/wiki/James_Clerk_Maxwell
19. http://en.wikipedia.org/wiki/Gauge_Transformation
20. http://en.wikipedia.org/wiki/Coulomb_gauge
21. *COMSOL AC/DC Module User's Guide*, Version 3.4, October 2007, COSMOL AB, Stockholm, Sweden, p. 92

Exercises

1. Build, mesh, and solve the 3D thin layer resistance model, thin layer approximation problem presented in this chapter.
2. Build, mesh, and solve the 3D thin layer resistance model, thin layer subdomain problem presented in this chapter.
3. Build, mesh, and solve the 3D electrostatic potential between two cylinders problem presented in this chapter.
4. Build, mesh, and solve the 3D electrostatic potential between five cylinders problem presented in this chapter.
5. Build, mesh, and solve the 3D magnetic field of a Helmholtz coil (static) problem presented in this chapter.
6. Build, mesh, and solve the 3D magnetic field of a Helmholtz coil with a magnetic test object problem presented in this chapter.
7. Explore other materials as applied in the 3D thin layer resistance models.
8. Explore other materials as applied in the model of a 3D magnetic field of a Helmholtz coil with a magnetic test object.
9. Explore adding more and/or different magnetic test objects to the 3D magnetic field of a Helmholtz coil model.
10. Explore the different geometries in the 3D thin layer resistance models.

9 Perfectly Matched Layer Models

In This Chapter

■ Perfectly Matched Layer Modeling Guidelines and Coordinate Considerations

PML Theory

One of the underlying fundamental difficulties in electromagnetic wave equation calculations (Maxwell's equations[1]) is dealing with a propagating wave interacting with boundaries (reflections). If the boundary of the model domain is terminated in the typical fashion,[2] unwanted reflections are typically incorporated into the solution. Fortunately, there is a methodology that works sufficiently well to essentially eliminate reflection problems at the domain boundary. That methodology is the perfectly matched layer (PML).

The PML is an approximation methodology originally developed in 1994 by Jean-Pierre Berenger[3] for use with FDTD[4] (finite-difference time-domain) electromagnetic modeling calculations. The PML technique has since been adapted and applied to other calculational methodologies that have similar domain mediated needs (e.g., FEM and others).[5] This methodology can be applied to a large variety of diverse wave equation problems.[6] In this chapter, however, it is applied only to electromagnetic problems within the context of the COMSOL RF Module.[7]

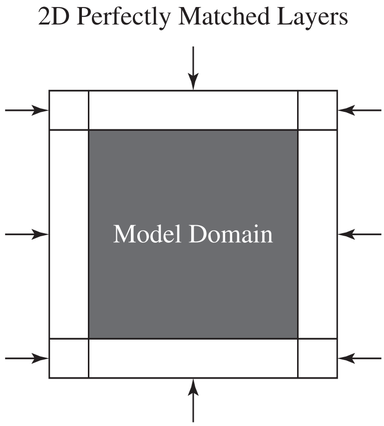


FIGURE 9.1 2D Cartesian domain with PMLs

NOTE For a broader detailed application of the PML methodology to other types of wave problems, refer to the literature.

In the COMSOL® Multiphysics® software, the PML technique is explicitly available in the RF Module, for application to electromagnetics modeling problems. The function of the PML methodology is to add anisotropic attenuating domains (layers) outside the modeled domain, so that the modeled domain has substantially reflectionless boundaries. Examples of modeling domains with PMLs include a 2D Cartesian domain with PMLs (Figure 9.1), a 3D Cartesian domain with PMLs (Figure 9.2), a 3D spherical domain with PMLs (Figure 9.3), and a 3D cylindrical domain with PMLs (Figure 9.4). The coordinate systems employed with the domain structures are those that are associated with their respective geometries.

NOTE To achieve the desired behavior of the wave equation PDE, the entire model domain, including the perfectly matched layers, is transformed to a complex coordinate system. For a Cartesian system (x, y, z), the transformation occurs as follows:

$$\frac{\partial}{\partial x} \rightarrow \frac{1}{1 + \dfrac{i\sigma(x)}{\omega}} \frac{\partial}{\partial x}; \quad \frac{\partial}{\partial y} \rightarrow \frac{1}{1 + \dfrac{i\sigma(y)}{\omega}} \frac{\partial}{\partial y}; \quad \frac{\partial}{\partial z} \rightarrow \frac{1}{1 + \dfrac{i\sigma(z)}{\omega}} \frac{\partial}{\partial z} \tag{9.1}$$

where $\sigma(x, y, z)$ is a step function that is zero inside the solution domain and a positive real number or an appropriate function of the designated coordinate variable (x, y, z) outside the solution domain and inside the PML.

FIGURE 9.2 3D Cartesian domain with PMLs

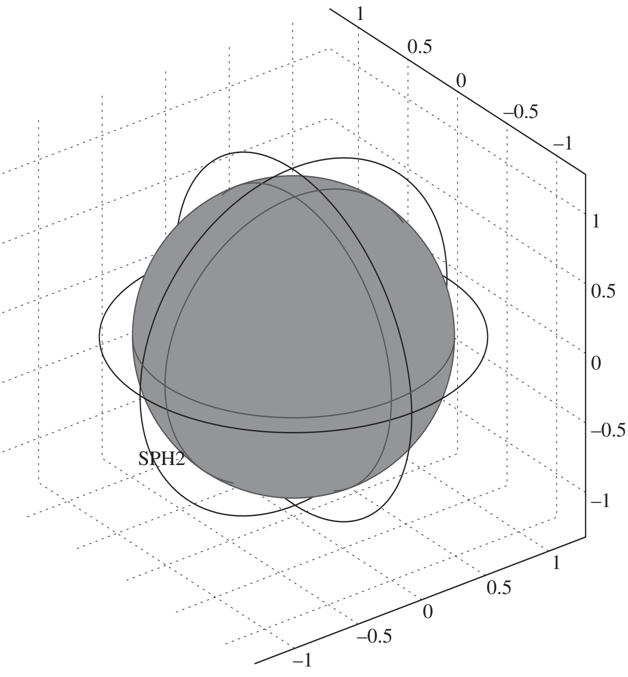


FIGURE 9.3 3D spherical domain with PMLs

FIGURE 9.4 3D cylindrical domain with PMLs

The transformation of the PDE in this fashion results in a solution with a multiplicative term that is, in general, as follows:

$$F(x, y, z) = f(x, y, z)*e^{-\frac{k\sigma(x, y, z)}{\omega}} \tag{9.2}$$

where $F(x, y, z) = f(x, y, z)*e^{-0}$ (the solution inside the domain)
$F(x, y, z) = f(x, y, z)*e^{-\kappa\sigma(x, y, z)/\omega}$ (the decaying solution within the PML domain)

At the outer PML boundary, the preferred boundary condition is the scattering boundary. However, if the attenuation of the propagating wave at the outer boundary of the PML is sufficient, then the particular boundary condition invoked is largely irrelevant. In such a case, the amplitude of the reflected wave will be sufficiently small as not to contribute to the final solution.

FIGURE 9.5 Wave equation solution example inside the modeling domain

Figures 9.5 and 9.6 show examples of wave equation solutions. For an example of the wave equation solution inside the modeling domain, see Figure 9.5. For an example of the wave equation solution inside the PML domain, see Figure 9.6.

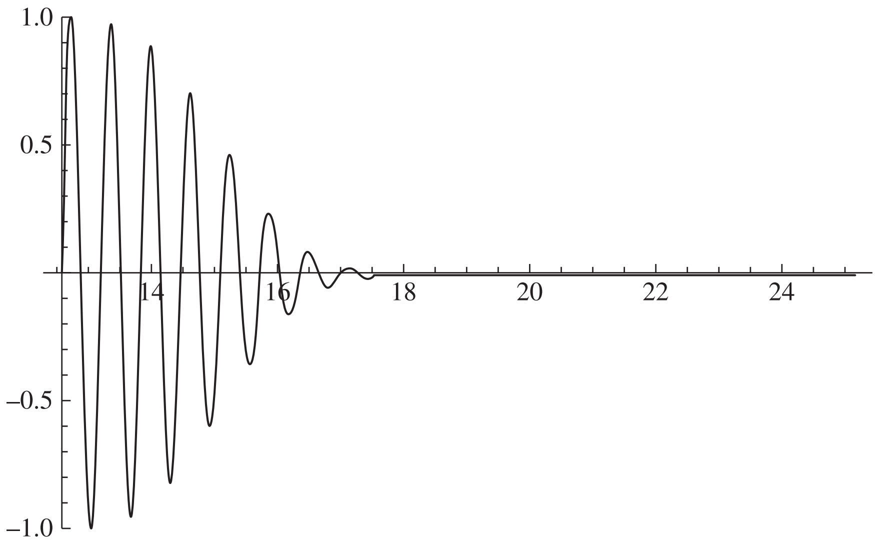


FIGURE 9.6 Wave equation solution example inside the PML domain

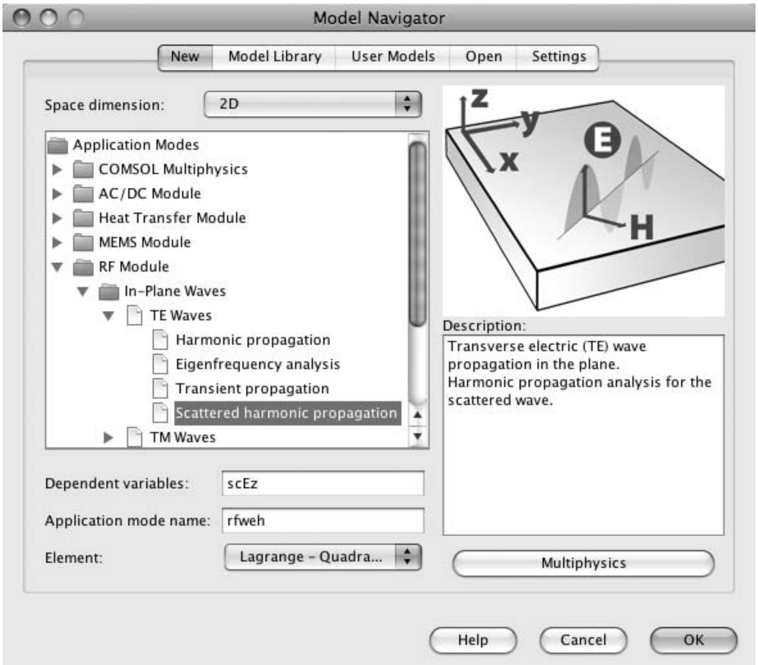


FIGURE 9.7 2D_PML_DL_1 Model Navigator setup

PML Models

2D Dielectric Lens Model, with PMLs

NOTE The dielectric lens is a concept borrowed from optical physics. In this application, the principles of optics are applied to lower-frequency electromagnetic waves to focus the impinging wavefront into the region of a sensor. The act of focusing the wavefront effectively amplifies the magnitude of the impinging signal.

The following numerical solution model (2D_PML_DL_1 model) is derived from a model that was originally developed by COMSOL as an RF Module tutorial model for the demonstration of the PML methodology. That model was developed for distribution with the RF Module software as part of the COMSOL RF Module Model Library.

To start building the 2D_PML_DL_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select RF Module > In-Plane Waves > TE Waves > Scattered harmonic propagation. See Figure 9.7. Click OK.

Table 9.1 Geometry Components

Name	Width	Height	Base	X	Y	Figure Number
R1	2.4	0.2	Corner	−1.2	1.0	9.8
R2	2.4	0.2	Corner	−1.2	−1.2	9.9
R3	0.2	2.4	Corner	−1.2	−1.2	9.10
R3	0.2	2.4	Corner	1.0	−1.2	9.11

NOTE The Model Navigator command sequence (In-Plane Waves > TE Waves > Scattered harmonic propagation) selects a transverse electric field (z-direction) wave traveling in the plane (x, y-plane) of the modeling domain.

Geometry Modeling

Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 9.1. Click OK after filling in the parameters of each separate rectangle in the Rectangle edit window. See Figures 9.8–9.11.

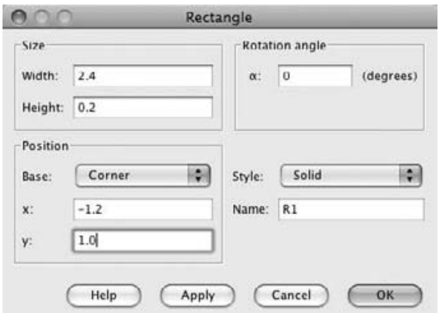


FIGURE 9.8 2D_PML_DL_1 model Rectangle (R1) edit window

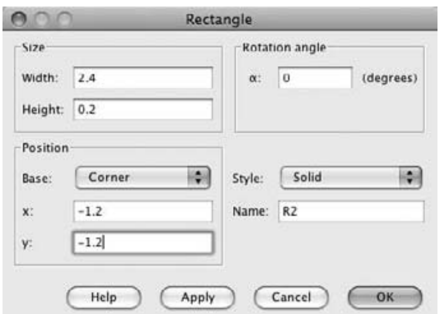


FIGURE 9.9 2D_PML_DL_1 model Rectangle (R2) edit window

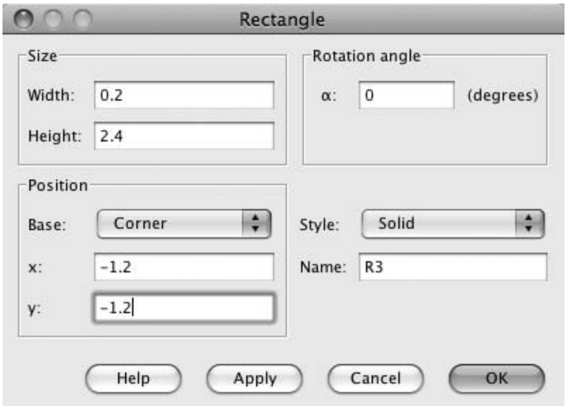


FIGURE 9.10 2D_PML_DL_1 model Rectangle (R3) edit window

Click the Zoom Extents button before drawing the next rectangle. Figure 9.12 shows the PML rectangles of model 2D_PML_DL_1.

Select File > Save As. Enter 2D_PML_DL_1.mph in the Save As edit window. See Figure 9.13. Click the Save button.

Using the menu bar, select Draw > Specify Objects > Circle. Enter Radius = 0.5, Base = Center, x = −0.5, and y = 0. See Figure 9.14. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter Width = 0.5, Height = 1.0, Base = Corner, x= −1.0, and y = −0.5. See Figure 9.15. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter C1-R5 in the Set formula edit window. See Figure 9.16.

FIGURE 9.11 2D_PML_DL_1 model Rectangle (R4) edit window

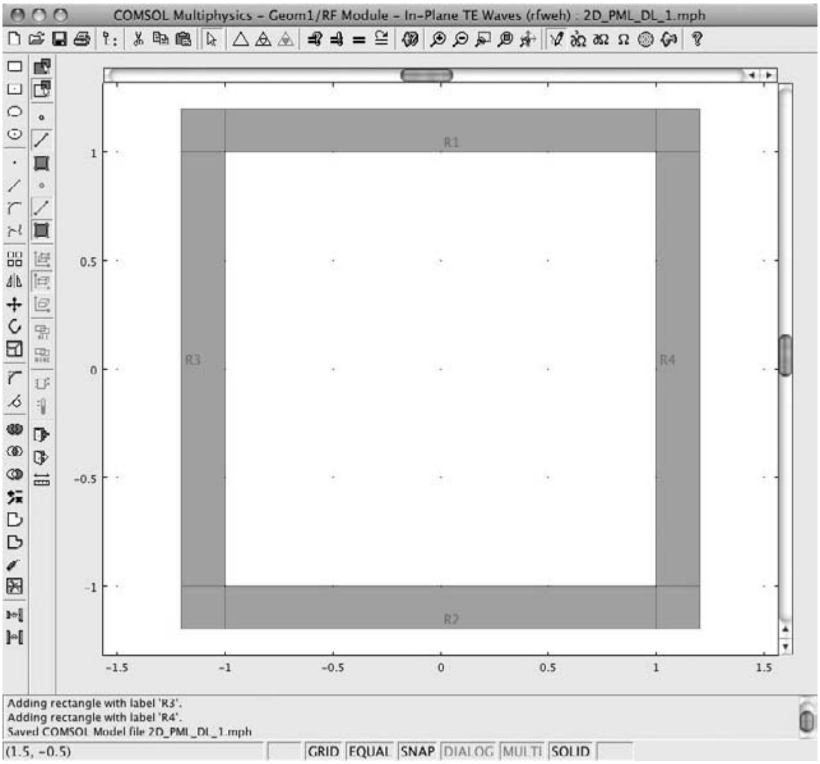


FIGURE 9.12 2D_PML_DL_1 model PML rectangles

FIGURE 9.13 2D_PML_DL_1 model Save As edit window

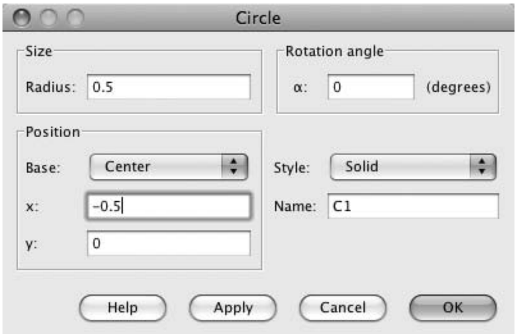


FIGURE 9.14 2D_PML_DL_1 model Circle (C1) edit window

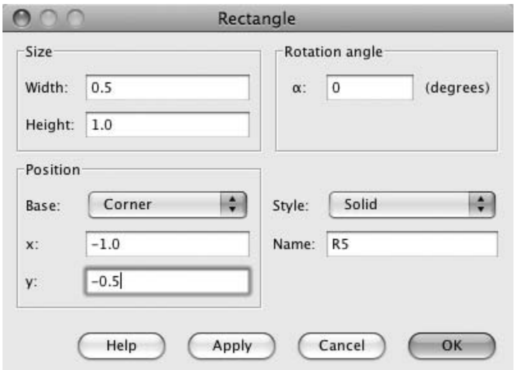


FIGURE 9.15 2D_PML_DL_1 model Rectangle (R5) edit window

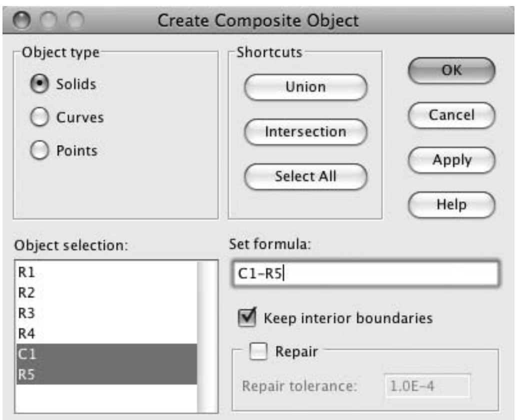


FIGURE 9.16 2D_PML_DL_1 model Create Composite Object edit window

FIGURE 9.17 2D_PML_DL_1 model dielectric lens (CO1)

Click OK. See Figure 9.17.

Using the menu bar, select Draw > Specify Objects > Square. Enter Width = 2, Base = Center, x = 0, and y = 0. See Figure 9.18. Click OK.

Figure 9.19 shows the model domain plus PMLs. Having established the geometry for the 2D_PML_DL_1 model, the next step is to define the fundamental Physics properties.

FIGURE 9.18 2D_PML_DL_1 model Square (SQ1) edit window

FIGURE 9.19 2D_PML_DL_1 model domain plus PMLs

Physics Application Mode Properties: In-Plane TE Waves (rfweh)

Select Physics > Properties. Select "Free space wavelength" from the Specify wave using pull-down list. See Figure 9.20. Click OK.

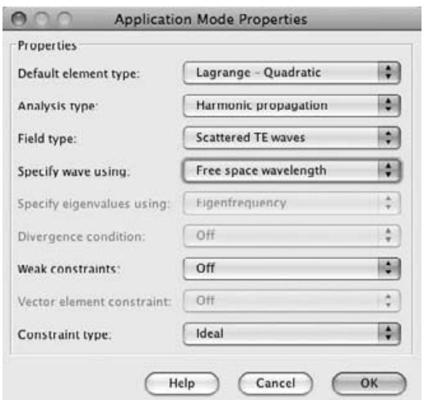


FIGURE 9.20 2D_PML_DL_1 model Application Mode Properties edit window

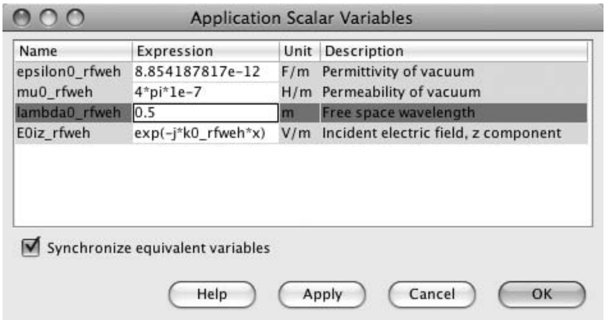


FIGURE 9.21 2D_PML_DL_1 model Application Scalar Variables (lambda0_rfweh) edit window

Physics Application Scalar Variables: In-Plane TE Waves (rfweh)

Select Physics > Application Scalar Variables. Enter 0.5 in the lambda0_rfweh edit window. See Figure 9.21. Click OK.

Physics Subdomain Settings: In-Plane TE Waves (rfweh)

Having established the basic Physics settings for the 2D_PML_DL_1 model, the next step is to define the fundamental Physics subdomain settings. Select Physics > Subdomain Settings. Click the PML tab. Select subdomains 1–4, 6, and 8–10 (the PMLs). Select "Cartesian" from the Type of PML pull-down list. Click the Apply button. See Figure 9.22.

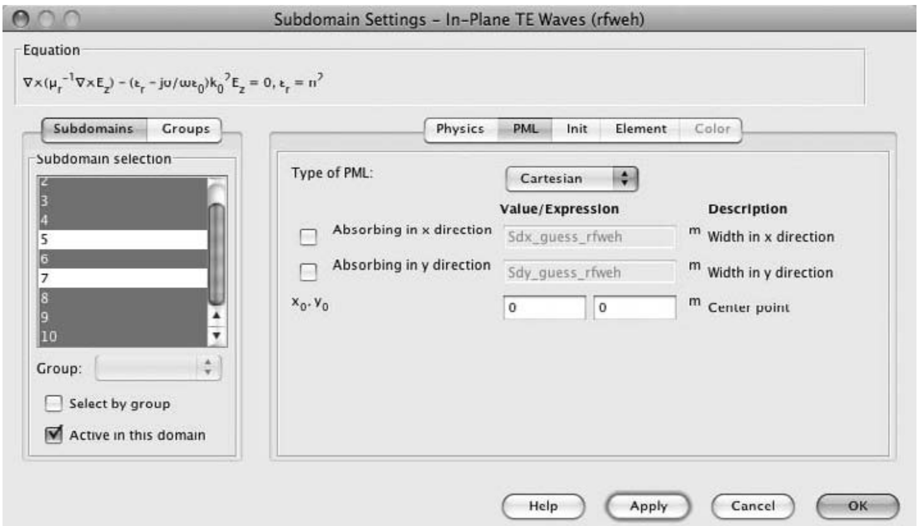


FIGURE 9.22 2D_PML_DL_1 model Subdomain Settings, PML type selection

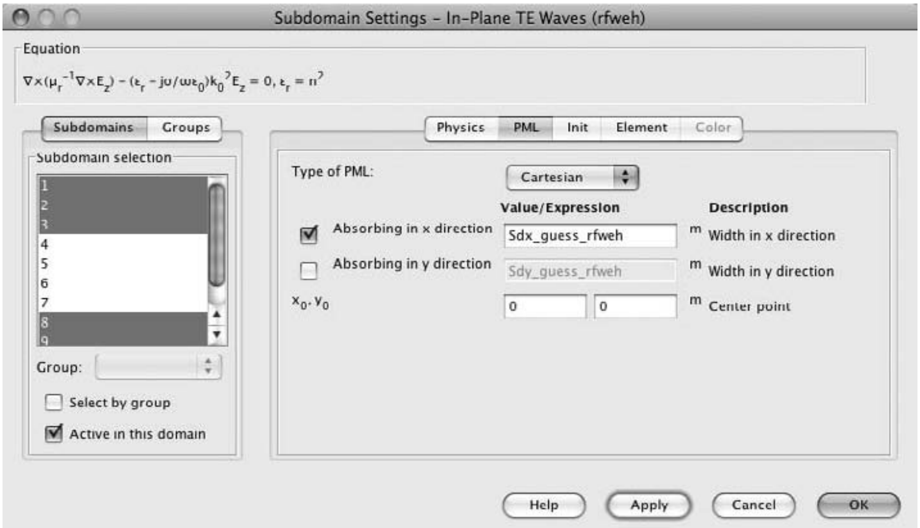


FIGURE 9.23 2D_PML_DL_1 model Subdomain Settings, x absorption

Select subdomains 1–3, and 8–10 (the vertical PMLs). Check the Absorbing in x direction check box. Click the Apply button. See Figure 9.23.

Select subdomains 1, 3, 4, 6, 8, and 10 (the horizontal PMLs). Check the Absorbing in y direction check box. Click the Apply button. See Figure 9.24.

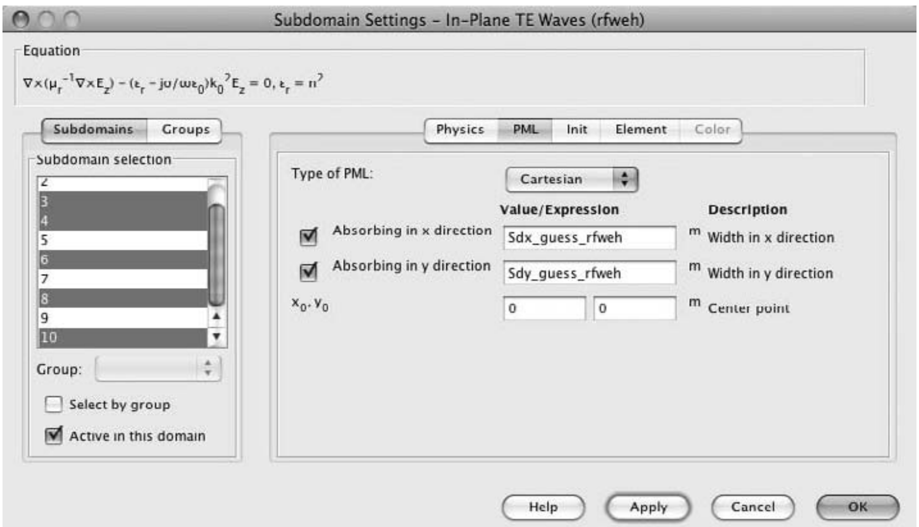


FIGURE 9.24 2D_PML_DL_1 model Subdomain Settings, y absorption

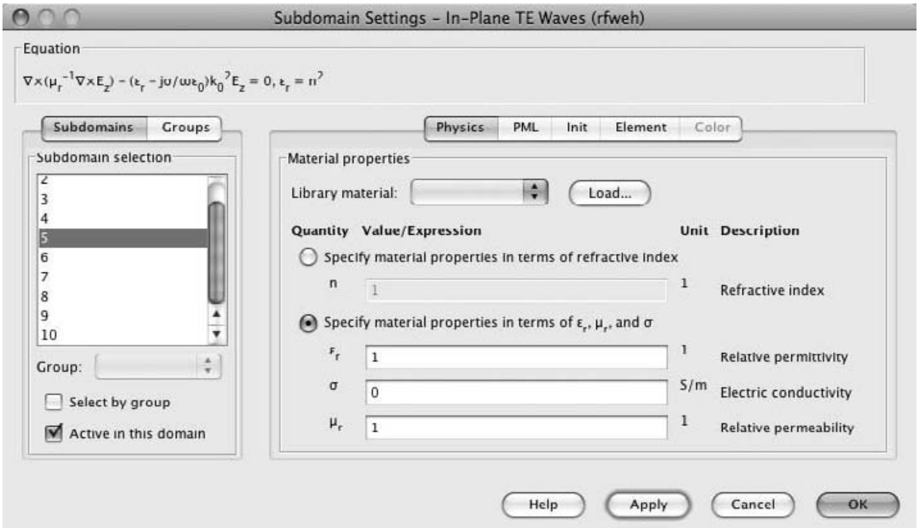


FIGURE 9.25 2D_PML_DL_1 model Subdomain Settings, Physics tab, subdomain 5

Click the Physics tab. Select subdomain 5 (the model domain). Enter $\varepsilon_r = 1$, $\sigma = 0$, and $\mu_r = 1$. Click the Apply button. See Figure 9.25.

Select subdomain 7 (the dielectric lens). Enter $\varepsilon_r = 3$, $\sigma = 0$, and $\mu_r = 1$. Click the Apply button. See Figure 9.26. Click OK.

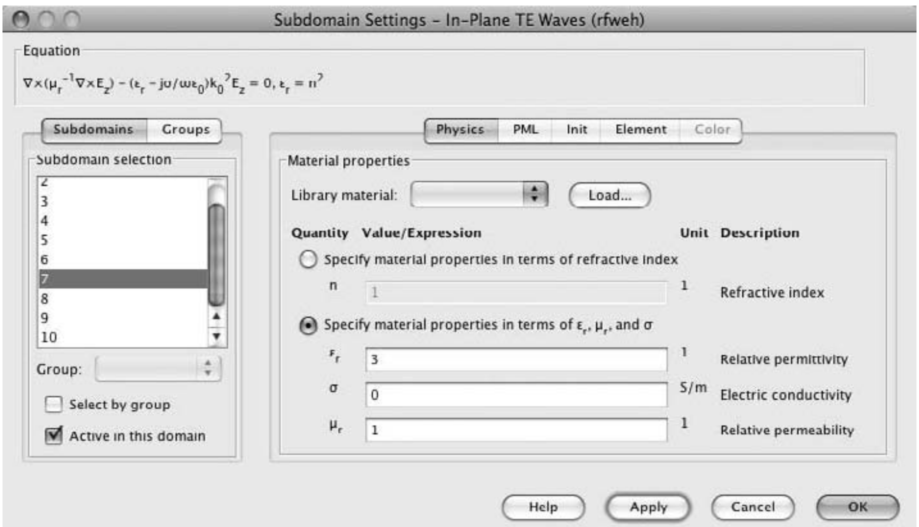


FIGURE 9.26 2D_PML_DL_1 model Subdomain Settings, Physics tab, subdomain 7

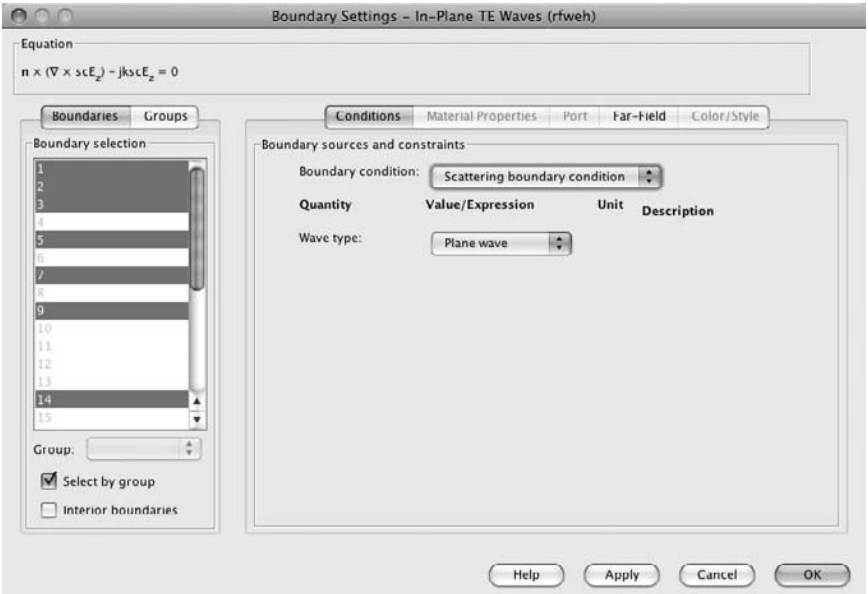


FIGURE 9.27 2D_PML_DL_1 model Boundary Settings

Physics Boundary Settings: In-Plane TE Waves (rfweh)

Having established the subdomain settings for the 2D_PML_DL_1 model, the next step is to define the fundamental Physics boundary settings. Using the menu bar, select Physics > Boundary Settings. Select boundary 1. Check the Select by Group check box to select all the outer edges of the PMLs (boundaries). Select "Scattering boundary condition" from the Boundary condition pull-down list. See Figure 9.27. Click OK.

Mesh Generation

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 7 (the dielectric lens). Enter 0.05 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 9.28.

Click the Mesh Selected button. Click the Select Remaining button. Click the Mesh Selected button. Click OK. See Figure 9.29.

Solving the 2D_PML_DL_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Parametric" in the Solver list. Enter lambda0_rfweh in the Parameter name edit window. Enter linspace(0.5,1.5,11) in the Parameter values edit window. (For later versions of the COMSOL Multiphysics software, enter range(0.5,1/10,1.5) in the Parameter values edit window.) See Figure 9.30.

Click OK. Using the menu bar, select Solve > Solve Problem.

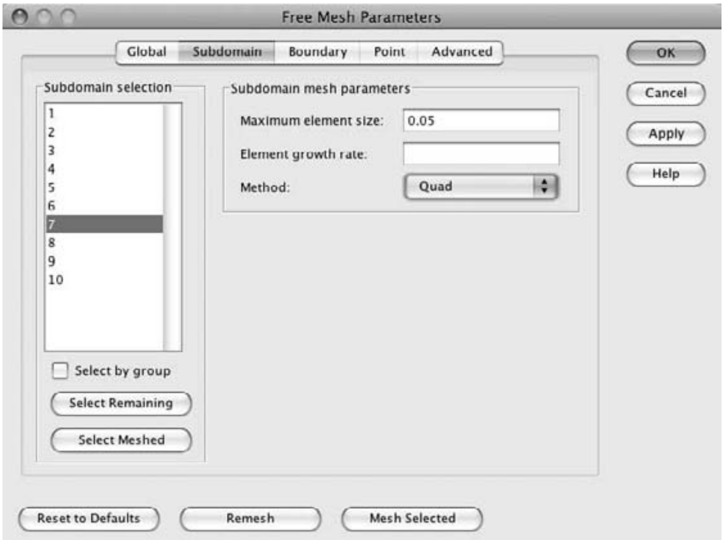


FIGURE 9.28 2D_PML_DL_1 model subdomain Free Mesh Parameters

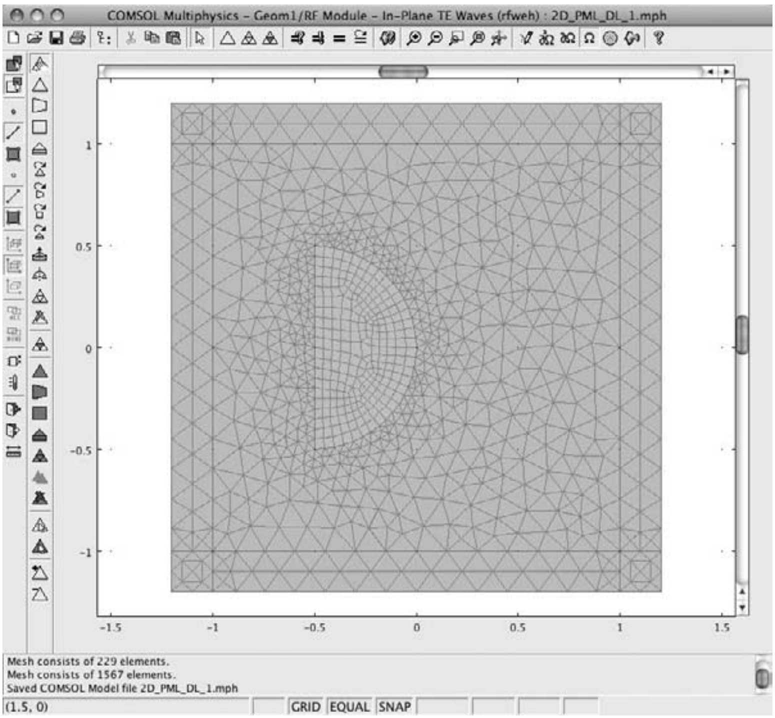


FIGURE 9.29 2D_PML_DL_1 model mesh

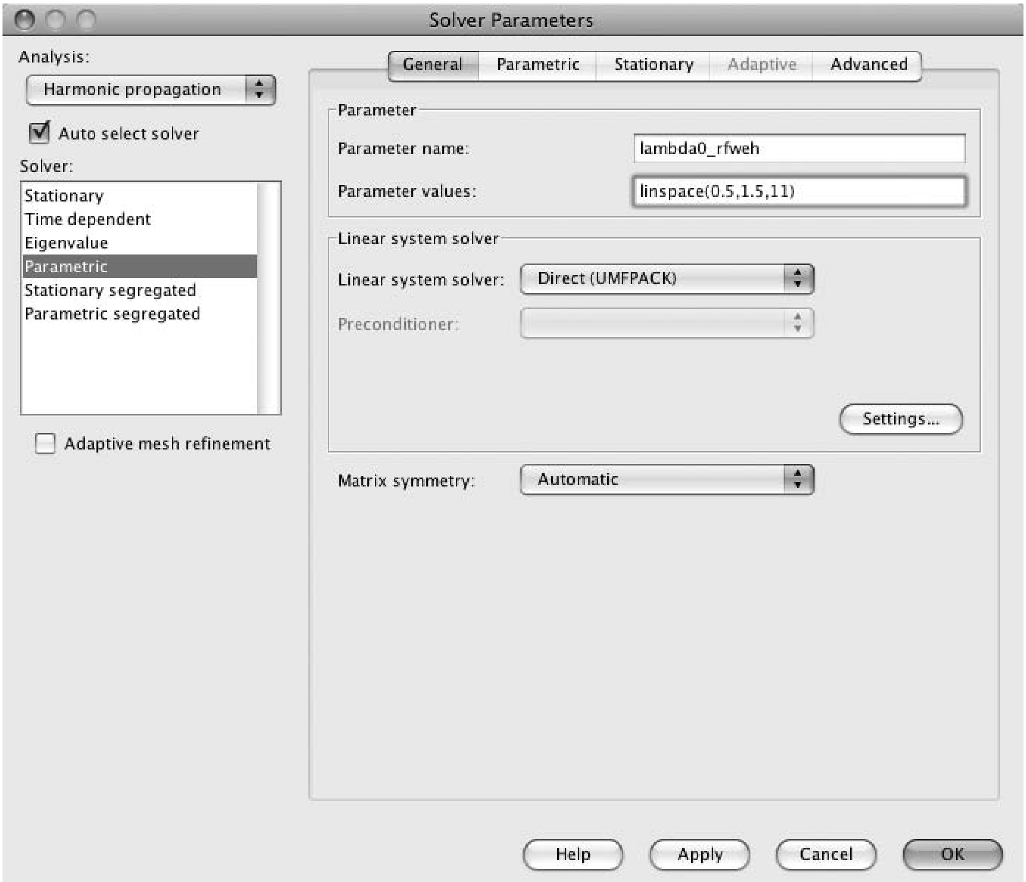


FIGURE 9.30 2D_PML_DL_1 model Solver Parameters edit window

Postprocessing and Visualization

The default plot shows a surface plot of the scattered electric field, *z*-component (V/m). See Figure 9.31.

An alternative approach to viewing the effect of the dielectric lens on the plane wave within the modeling domain is to suppress the plot within the PMLs and visualize the electric field, *z*-component. Using the menu bar, select Options > Suppress > Suppress Subdomains. Select subdomains 1–4, 6, and 8–10. Click the Apply button. See Figure 9.32. Click OK.

Using the menu bar, select Postprocessing > Plot Parameters > Surface. Select “Electric field, z component” from the Predefined quantities pull-down list. See Figure 9.33.

FIGURE 9.31 2D_PML_DL_1 model solution, scattered electric field, *z*-component (V/m)

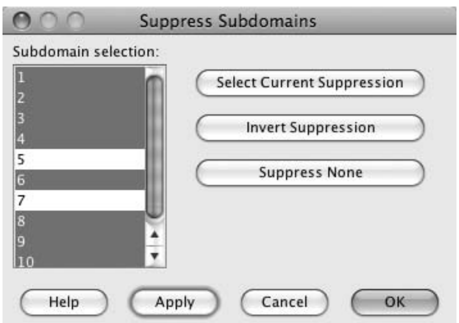


FIGURE 9.32 2D_PML_DL_1 model Suppress Subdomains

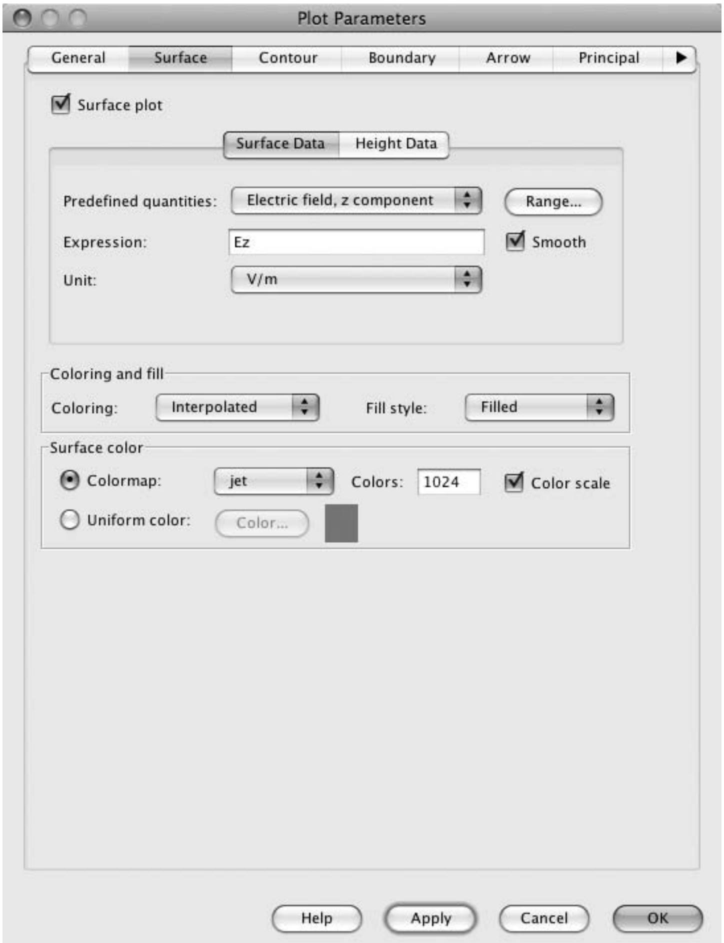


FIGURE 9.33 2D_PML_DL_1 model postprocessing Plot Parameters, Surface tab

Click OK. See Figure 9.34.

Using the menu bar, select Postprocessing > Plot Parameters > Animate. Verify that all the Solutions to use are selected. See Figure 9.35. Click the Start Animation button.

2D Dielectric Lens Model, with PMLs: Summary and Conclusions

The 2D dielectric lens model, with PMLs (2D_PML_DL_1 model), has been built and solved. This model employs PMLs and a dielectric lens to explore the geometric behavior of transverse electric field RF waves in the presence of a focusing element. It can easily be observed by watching the animation that the position and

FIGURE 9.34 2D_PML_DL_1 model electric field, *z*-component

intensity of the electric field, *z*-component varies greatly as a function of the free space wavelength.

2D Dielectric Lens Model, without PMLs

The following numerical solution model (2D_NoPML_DL_1 model) is derived from the immediately preceding model in this chapter (2D_PML_DL_1). The purpose in building this model is to empirically demonstrate the differences that are seen when PMLs are not employed.

To start building the 2D_NoPML_DL_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select RF Module > In-Plane Waves > TE Waves > Scattered harmonic propagation. See Figure 9.36. Click OK.

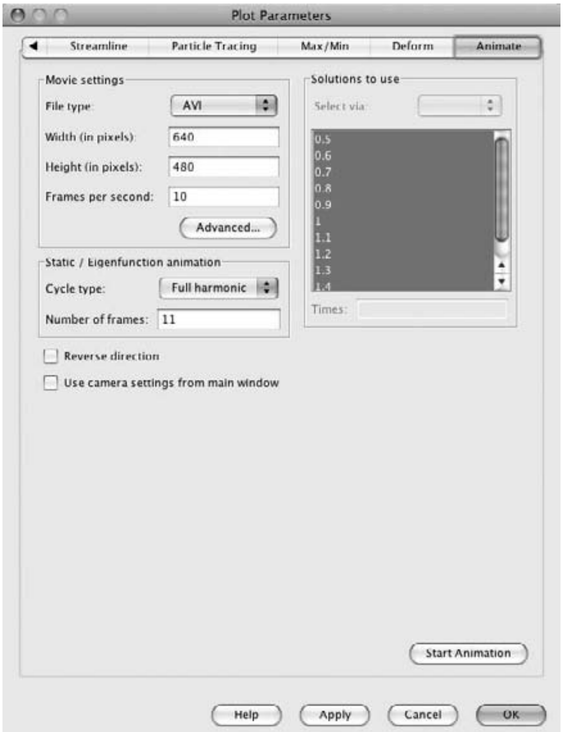


FIGURE 9.35 2D_PML_DL_1 model Plot Parameters, Animate tab

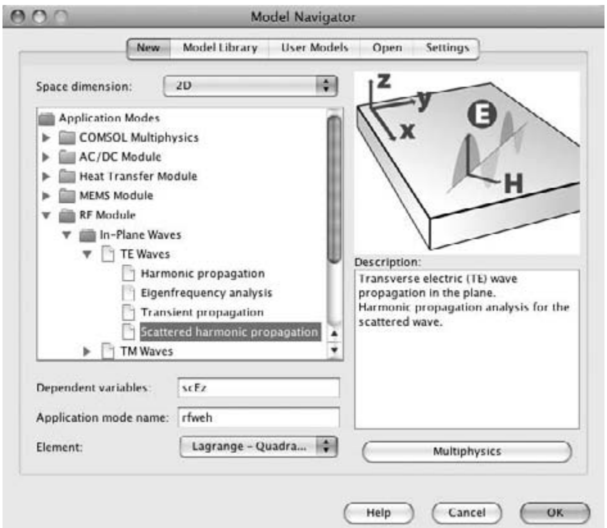


FIGURE 9.36 2D_NoPML_DL_1 Model Navigator setup

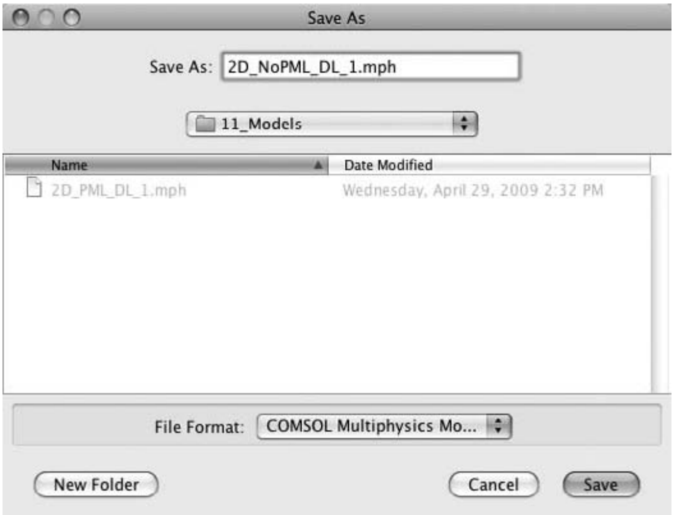


FIGURE 9.37 2D_NoPML_DL_1 model Save As edit window

NOTE The Model Navigator command sequence (In-Plane Waves > TE Waves > Scattered harmonic propagation) selects a transverse electric field (z-direction) wave traveling in the plane (x, y-plane) of the modeling domain.

Select File > Save As. Enter 2D_NoPML_DL_1.mph in the Save As edit window. See Figure 9.37. Click the Save button.

Geometry Modeling

Using the menu bar, select Draw > Specify Objects > Circle. Enter Radius = 0.5, Base = Center, x = −0.5, and y = 0. See Figure 9.38. Click OK.

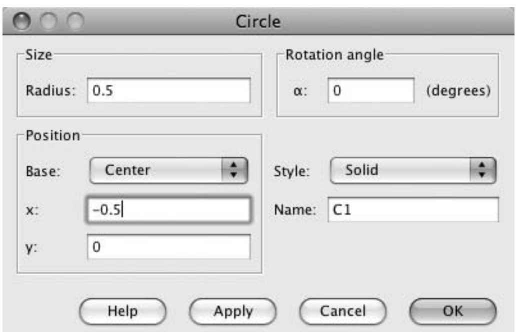


FIGURE 9.38 2D_NoPML_DL_1 model Circle (C1) edit window

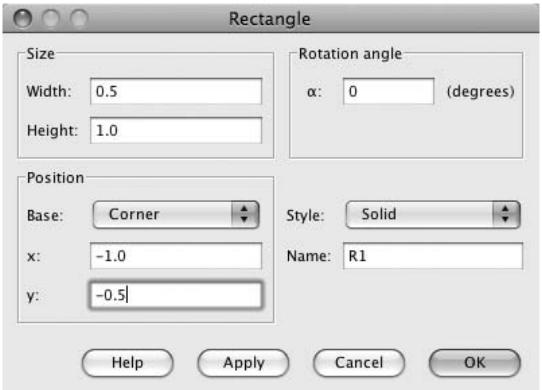


FIGURE 9.39 2D_NoPML_DL_1 model Rectangle (R1) edit window

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter Width = 0.5, Height = 1.0, Base = Corner, x = −1.0, and y = −0.5. See Figure 9.39. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter C1-R1 in the Set formula edit window. See Figure 9.40.

Click OK. See Figure 9.41.

Using the menu bar, select Draw > Specify Objects > Square. Enter Width = 2, Base = Center, x = 0, and y = 0. See Figure 9.42.

Click OK, and then click the Zoom Extents button. See Figure 9.43.

Having established the geometry for the 2D_NoPML_DL_1 model, the next step is to define the fundamental Physics properties.

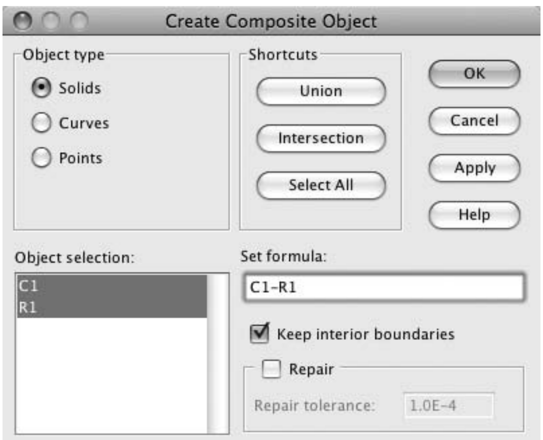


FIGURE 9.40 2D_NoPML_DL_1 model Create Composite Object edit window

FIGURE 9.41 2D_NoPML_DL_1 model dielectric lens (CO1)

FIGURE 9.42 2D_NoPML_DL_1 model Square (SQ1) edit window

FIGURE 9.43 2D_NoPML_DL_1 model domain

Physics Application Mode Properties: In-Plane TE Waves (rfweh)

Select Physics > Properties. Select “Free space wavelength” from the Specify wave using pull-down list. See Figure 9.44. Click OK.

FIGURE 9.44 2D_NoPML_DL_1 model Application Mode Properties edit window

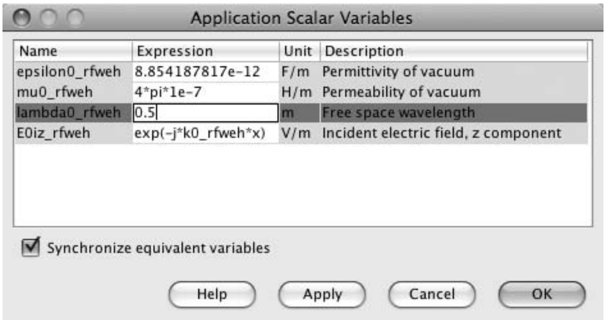


FIGURE 9.45 2D_NoPML_DL_1 model Application Scalar Variables edit window (lambda0_rfweh)

Physics Application Scalar Variables: In-Plane TE Waves (rfweh)

Select Physics > Scalar Variables. Enter 0.5 in the lambda0_rfweh edit window. See Figure 9.45. Click OK.

Physics Subdomain Settings: In-Plane TE Waves (rfweh)

Having established the basic Physics settings for the 2D_NoPML_DL_1 model, the next step is to define the fundamental Physics subdomain settings. Select Physics > Subdomain Settings. Select subdomain 1 (the model domain). Enter $\varepsilon_r = 1$, $\sigma = 0$, and $\mu_r = 1$. Click the Apply button. See Figure 9.46.

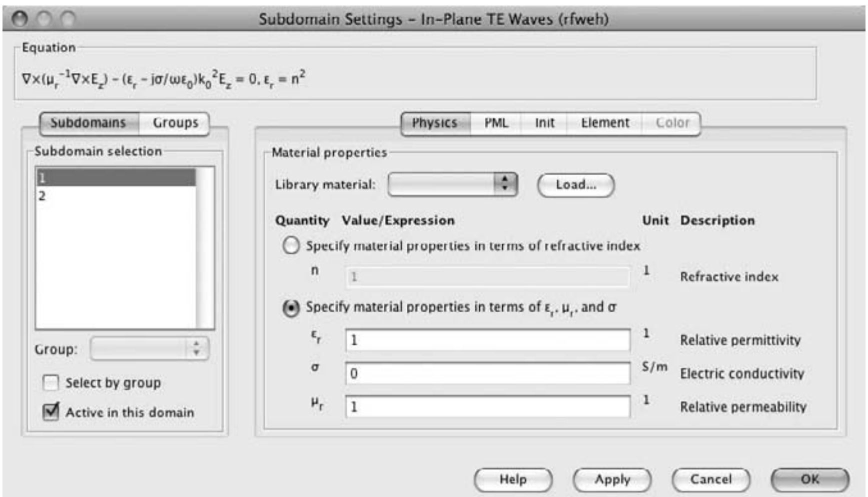


FIGURE 9.46 2D_NoPML_DL_1 model Subdomain Settings, subdomain 1

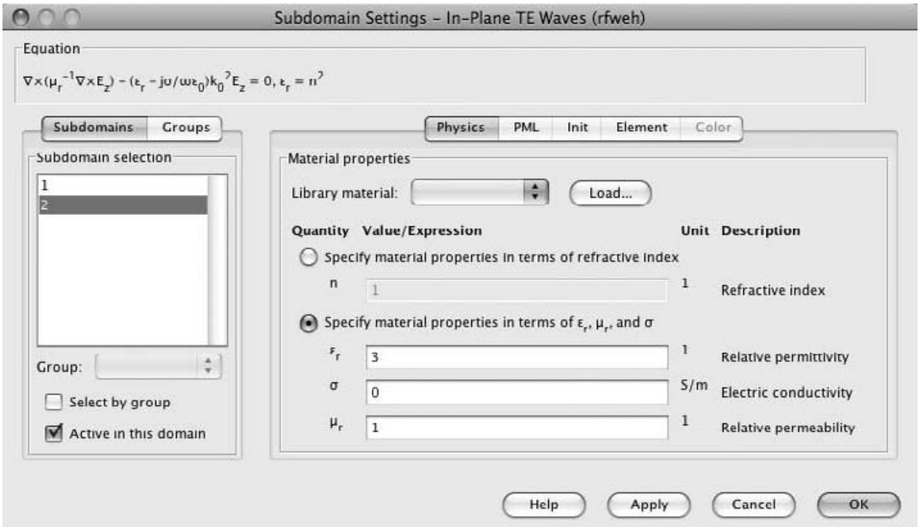


FIGURE 9.47 2D_NoPML_DL_1 model Subdomain Settings, subdomain 2

Select subdomain 2 (the dielectric lens). Enter $\varepsilon_r = 3$, $\sigma = 0$, and $\mu_r = 1$. Click the Apply button. See Figure 9.47. Click OK.

Physics Boundary Settings: In-Plane TE Waves (rfweh)

Having established the subdomain settings for the 2D_NoPML_DL_1 model, the next step is to define the fundamental Physics boundary settings. Using the menu bar, select Physics > Boundary Settings. Select boundaries 1, 2, 3, and 5 (the outer edges of the model domain). Select "Scattering boundary condition" from the Boundary condition pull-down list. See Figure 9.48. Click OK.

Mesh Generation

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 2 (the dielectric lens). Enter 0.05 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 9.49.

Click the Mesh Selected button. Click the Select Remaining button. Click the Mesh Selected button. Click OK. See Figure 9.50.

Solving the 2D_NoPML_DL_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Parametric" in the Solver list. Enter lambda0_rfweh in the Parameter name edit window. Enter linspace(0.5,1.5,11) in the Parameter values edit window. (For later versions of the COMSOL Multiphysics software enter range(0.5,1/10,1.5) in the Parameter values edit window.) See Figure 9.51.

FIGURE 9.48 2D_NoPML_DL_1 model Boundary Settings

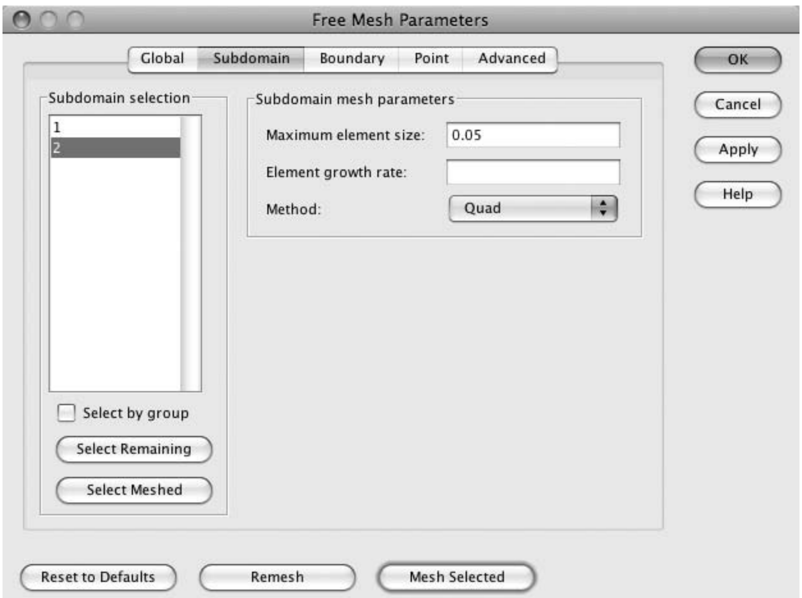


FIGURE 9.49 2D_NoPML_DL_1 model subdomain Free Mesh Parameters

FIGURE 9.50 2D_NoPML_DL_1 model mesh

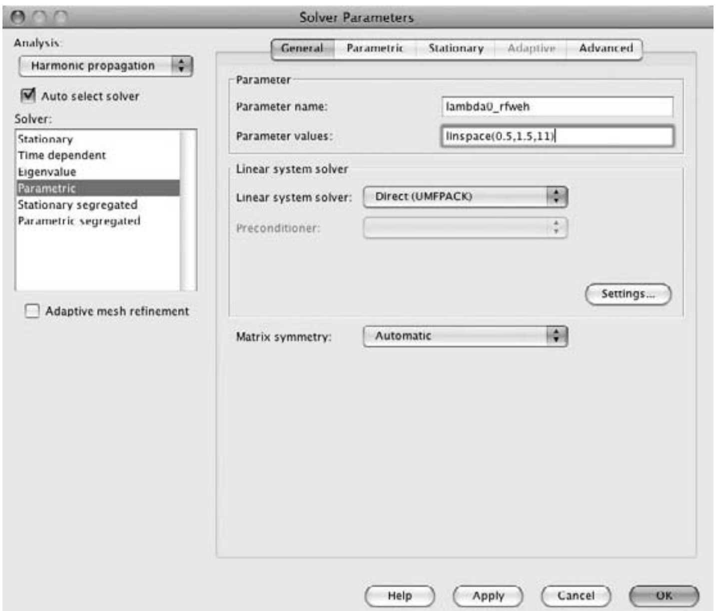


FIGURE 9.51 2D_NoPML_DL_1 model Solver Parameters edit window

FIGURE 9.52 2D_NoPML_DL_1 model solution, scattered electric field, z-component (V/m)

Click OK. Using the menu bar, select Solve > Solve Problem.

Postprocessing and Visualization

The default plot shows a surface plot of the scattered electric field, z-component (V/m). See Figure 9.52.

An alternative approach to viewing the effect of the dielectric lens on the plane wave within the modeling domain is to view the electric field, z-component. Using the menu bar, select Postprocessing > Plot Parameters > Surface. Select “Electric field, z component” from the Predefined quantities pull-down list. See Figure 9.53.

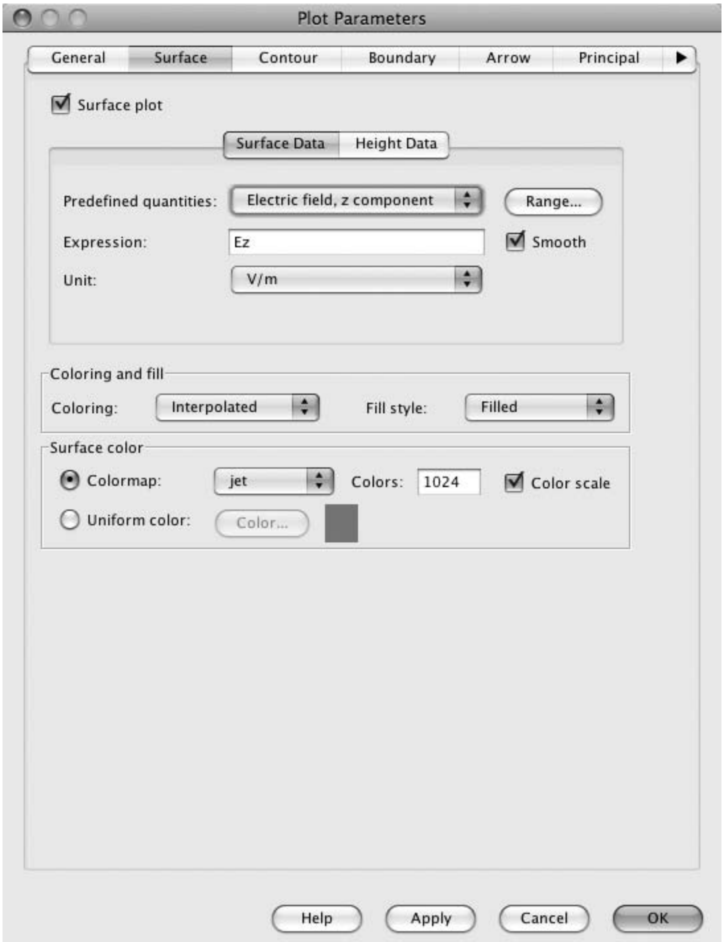


FIGURE 9.53 2D_NoPML_DL_1 model postprocessing Plot Parameters, Surface tab

Click OK. See Figure 9.54.

Using the menu bar, select Postprocessing > Plot Parameters > Animate. Verify that all the Solutions to use are selected. See Figure 9.55. Click the Start Animation button.

2D Dielectric Lens Model, with and without PMLs: Summary and Conclusions

The 2D dielectric lens models, with and without PMLs (2D_PML_DL_1 and 2D_NoPML_DL_1, respectively) have been built and solved. The best method of comparison between the two models is to view visualizations for the electric field,

FIGURE 9.54 2D_NoPML_DL_1 model electric field, *z*-component

z-component for the same wavelength from each model together. Figures 9.56 through 9.61 show visualizations for 0.5 m (Figures 9.56 and 9.57), 1.0 m (Figures 9.58 and 9.59), and 1.5 m (Figures 9.60 and 9.61).

The differences in the electric field, *z*-component visualizations between the PML and no-PML models amount to approximately 2%. Depending on the nature of the problem, such differences may or may not be significant. What these differences show the modeler is that he or she needs to understand the application environment well to build the best model. The PML model best approximates a free space environment (no reflections). For other environments, the modeler needs to determine the best boundary condition approximation using standard practices and a first principles approach. When all else fails (or even before), do a first principles analysis of the environment before building the model.

FIGURE 9.55 2D_NoPML_DL_1 model Plot Parameters, Animate tab

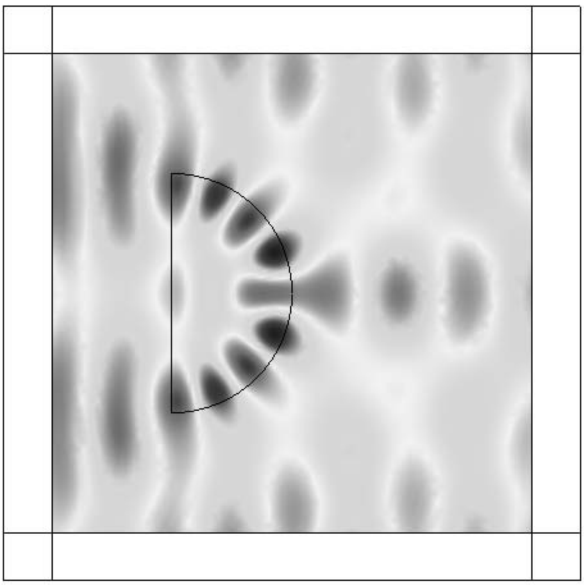


FIGURE 9.56 2D_PML_DL_1 model plot electric field, *z*-component, 0.5 m

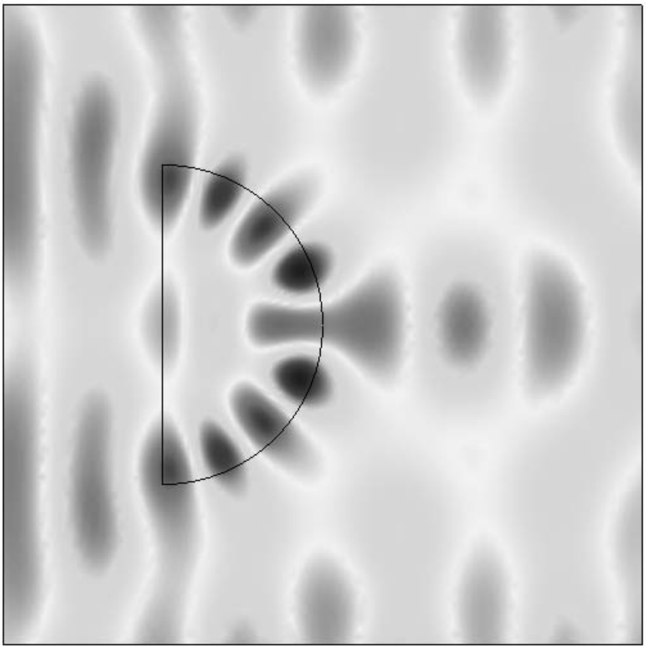


FIGURE 9.57 2D_NoPML_DL_1 model plot electric field, *z*-component, 0.5 m

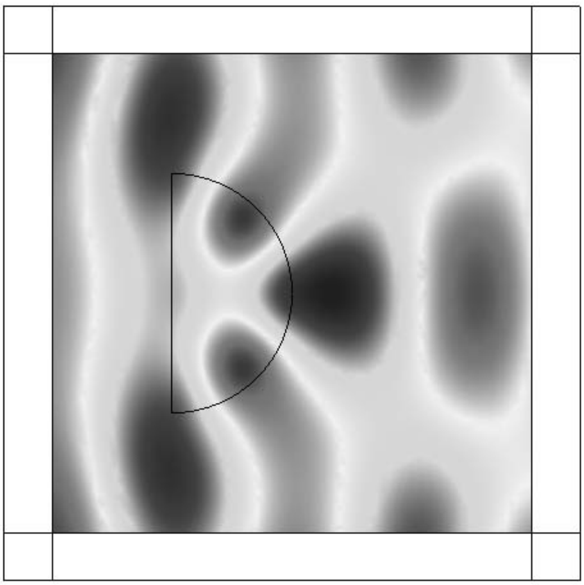


FIGURE 9.58 2D_PML_DL_1 model plot electric field, *z*-component, 1.0 m

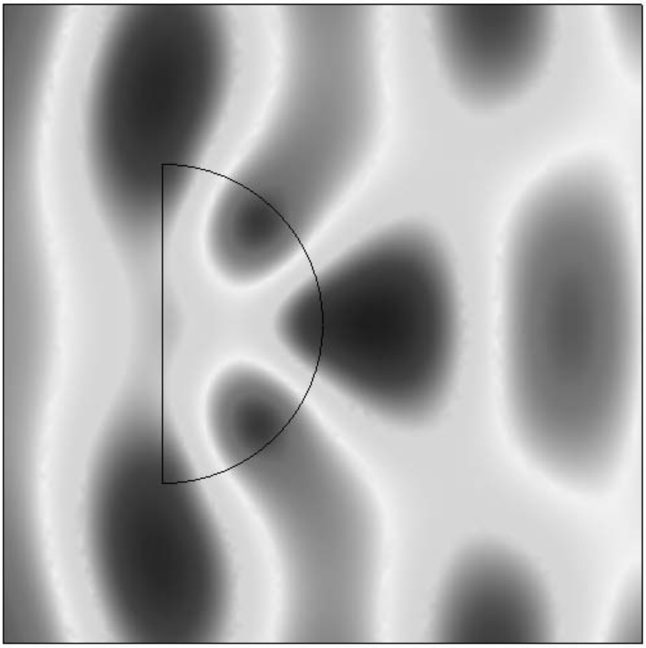


FIGURE 9.59 2D_NoPML_DL_1 model plot electric field, *z*-component, 1.0 m

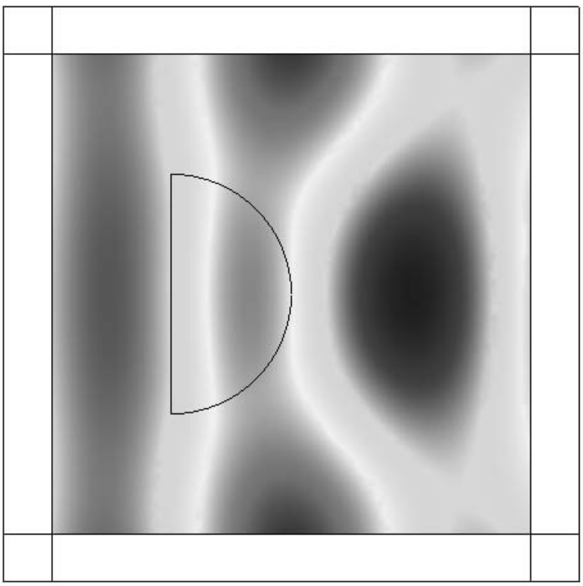


FIGURE 9.60 2D_PML_DL_1 model plot electric field, *z*-component, 1.5 m

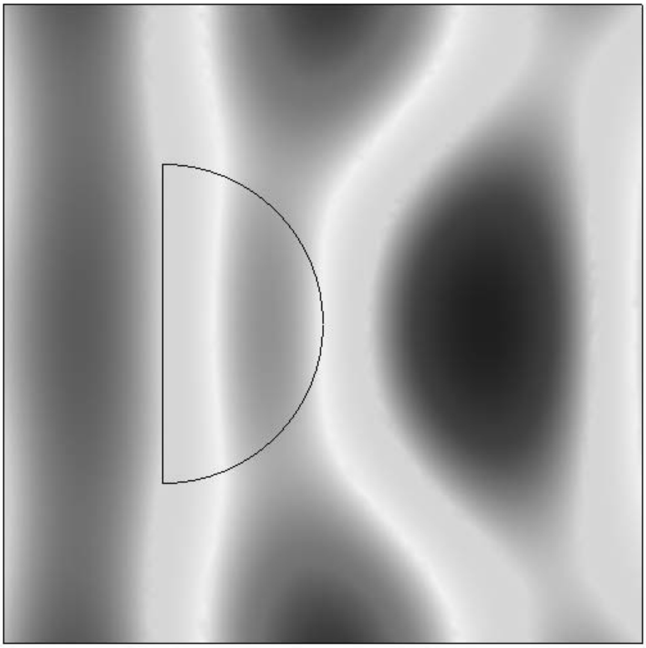


FIGURE 9.61 2D_NoPML_DL_1 model plot electric field, *z*-component, 1.5 m

2D Concave Mirror Model, with PMLs

The following numerical solution model (2D_PML_CM_1 model) is derived from the preceding dielectric lens model (2D_PML_DL_1 model). In this case, however, the electromagnetic waves interact with a fixed, curved metallic mirror. The purpose of this model (2D_PML_CM_1) and the following model (2D_NoPML_CM_1) is to demonstrate empirically the difference between having or not having PMLs at the model boundaries.

To start building the 2D_PML_CM_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select RF Module > In-Plane Waves > TE Waves > Scattered harmonic propagation. See Figure 9.62. Click OK.

NOTE The Model Navigator command sequence (In-Plane Waves > TE Waves > Scattered harmonic propagation) selects a transverse electric field (z-direction) wave traveling in the plane (x, y-plane) of the modeling domain.

Geometry Modeling

Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 9.2. Click OK after filling in the parameters

Table 9.2 Geometry Components

Name	Width	Height	Base	X	Y	Figure Number
R1	2.4	0.2	Corner	−1.2	1.0	9.63
R2	2.4	0.2	Corner	−1.2	−1.2	9.64
R3	0.2	2.4	Corner	−1.2	−1.2	9.65
R3	0.2	2.4	Corner	1.0	−1.2	9.66

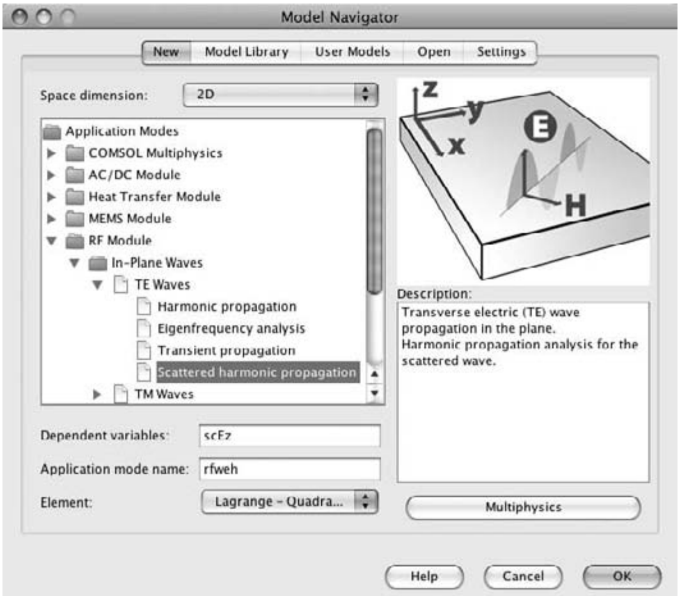


FIGURE 9.62 2D_PML_CM_1 Model Navigator setup

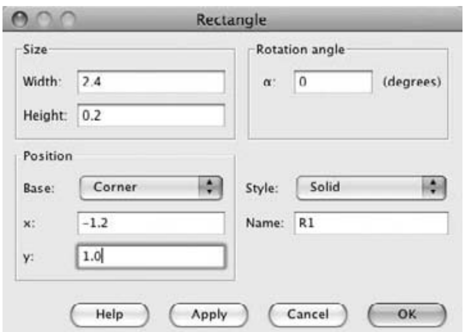


FIGURE 9.63 2D_PML_CM_1 model Rectangle (R1) edit window

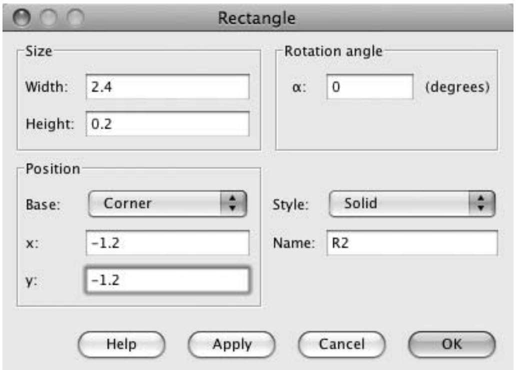


FIGURE 9.64 2D_PML_CM_1 model Rectangle (R2) edit window

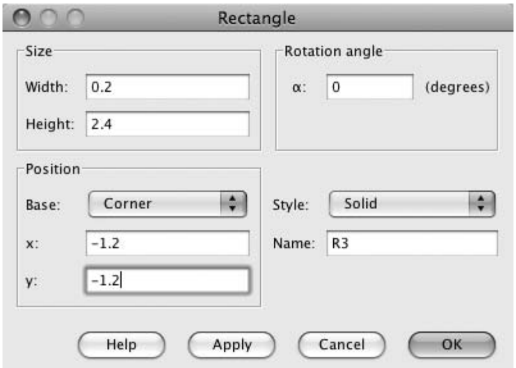


FIGURE 9.65 2D_PML_CM_1 model Rectangle (R3) edit window

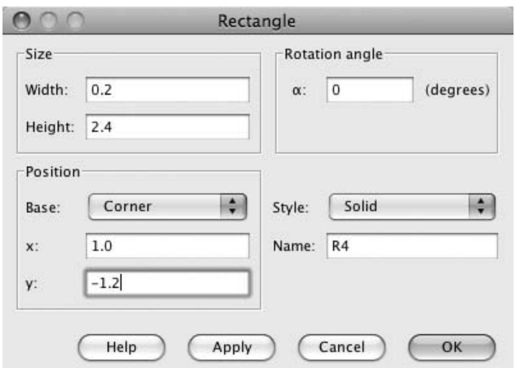


FIGURE 9.66 2D_PML_CM_1 model Rectangle (R4) edit window

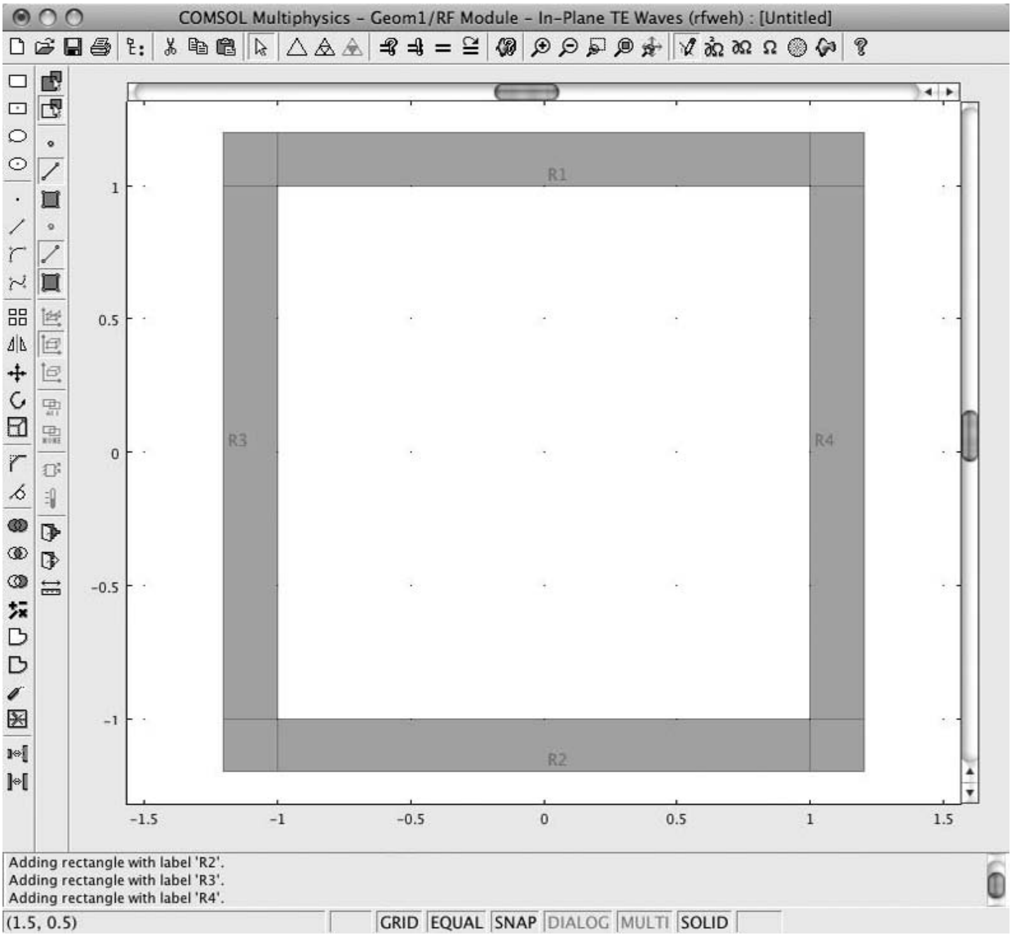


FIGURE 9.67 2D_PML_CM_1 model PML rectangles

of each separate rectangle in the Rectangle edit window. See Figures 9.63–9.66. Click the Zoom Extents button before drawing the next rectangle. Figure 9.67 shows the PML rectangles of model 2D_PML_CM_1.

Select File > Save As. Enter 2D_PML_CM_1.mph in the Save As edit window. See Figure 9.68. Click the Save button.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes = 0.3, B-semiaxes = 0.5, Base = Center, x = 0, and y = 0. See Figure 9.69. Click OK.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes = 0.29, B-semiaxes = 0.49, Base = Center, x = 0, and y = 0. See Figure 9.70. Click OK.

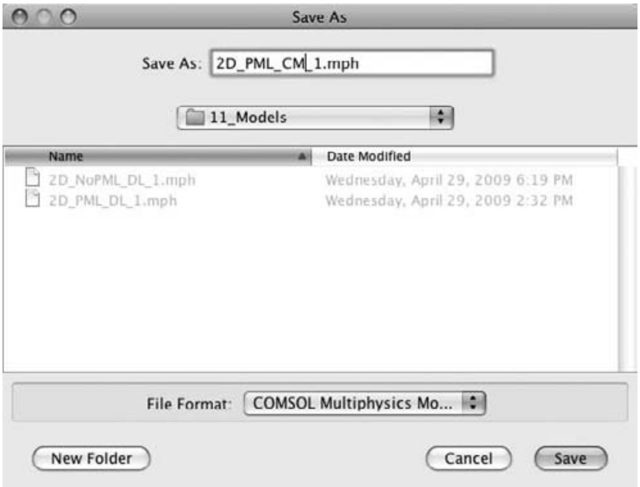


FIGURE 9.68 2D_PML_CM_1 model Save As edit window

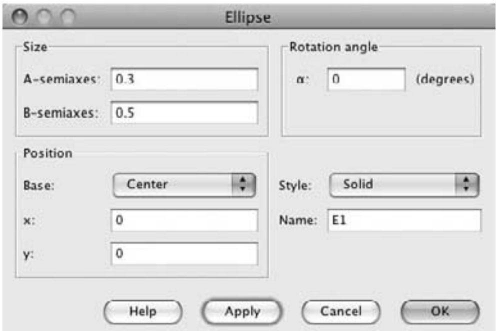


FIGURE 9.69 2D_PML_CM_1 model Ellipse (E1) edit window

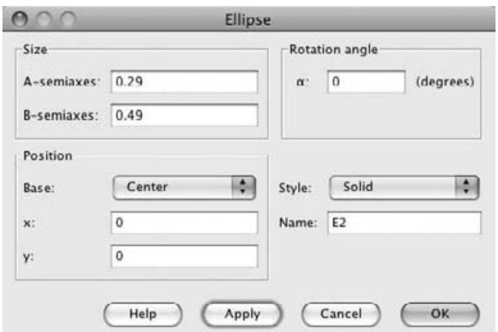


FIGURE 9.70 2D_PML_CM_1 model Ellipse (E2) edit window

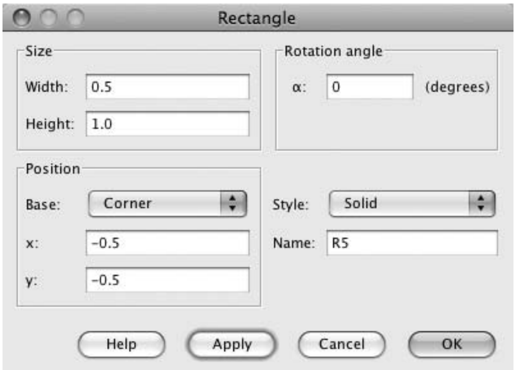


FIGURE 9.71 2D_PML_CM_1 model Rectangle (R5) edit window

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter Width = 0.5, Height = 1.0, Base = Corner, x = −0.5, and y = −0.5. See Figure 9.71. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter E1-E2-R5 in the Set formula edit window. See Figure 9.72.

Click OK. See Figure 9.73.

Using the menu bar, select Draw > Specify Objects > Square. Enter Width = 2.0, Base = Center, x = 0, and y = 0. See Figure 9.74. Click OK.

Using the menu bar, select Edit > Select All. See Figure 9.75.

Having established the geometry for the 2D_PML_CM_1 model, the next step is to define the fundamental Physics properties.

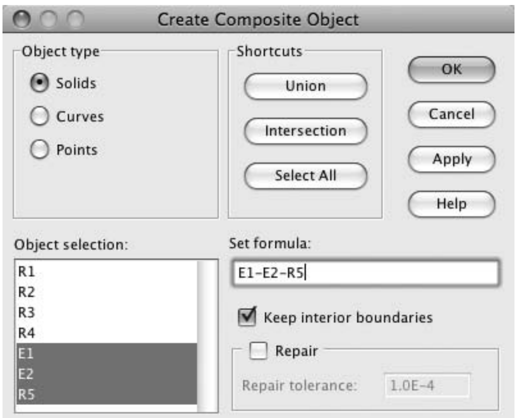


FIGURE 9.72 2D_PML_CM_1 model Create Composite Object edit window

FIGURE 9.73 2D_PML_CM_1 model concave mirror (CO1)

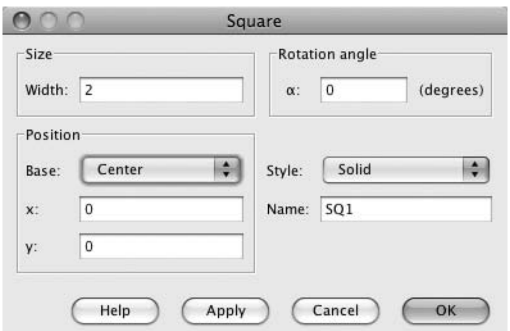


FIGURE 9.74 2D_PML_CM_1 model Square (SQ1) edit window

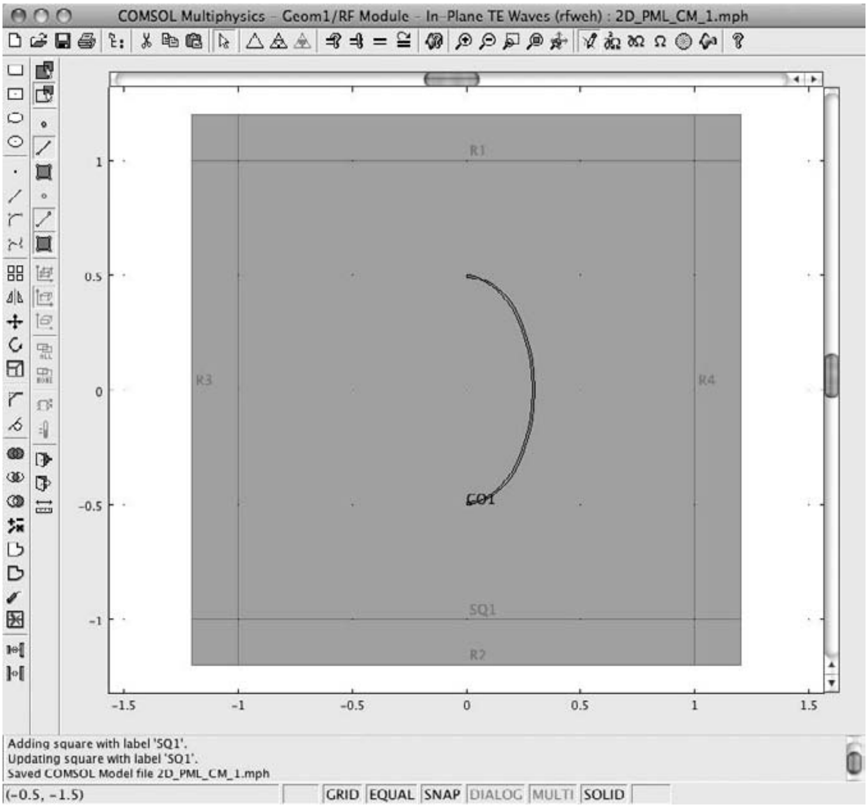


FIGURE 9.75 2D_PML_CM_1 model domain plus PMLs

Physics Application Mode Properties: In-Plane TE Waves (rfweh)

Select Physics > Properties. Select "Free space wavelength" from the Specify wave using pull-down list. See Figure 9.76. Click OK.

Physics Application Scalar Variables: In-Plane TE Waves (rfweh)

Select Physics > Scalar Variables. Enter 0.5 in the lambda0_rfweh edit window. See Figure 9.77. Click OK.

Physics Subdomain Settings: In-Plane TE Waves (rfweh)

Having established the basic Physics settings for the 2D_PML_CM_1 model, the next step is to define the fundamental Physics subdomain settings. Select Physics > Subdomain Settings. Click the PML tab. Select subdomains 1–4, 6, and 8–10 (the PMLs). Select "Cartesian" from the Type of PML pull-down list. Click the Apply button. See Figure 9.78.

FIGURE 9.76 2D_PML_CM_1 model Application Mode Properties edit window

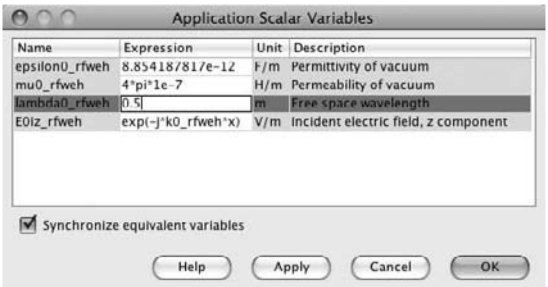


FIGURE 9.77 2D_PML_CM_1 model Application Scalar Variables (lambda0_rfweh) edit window

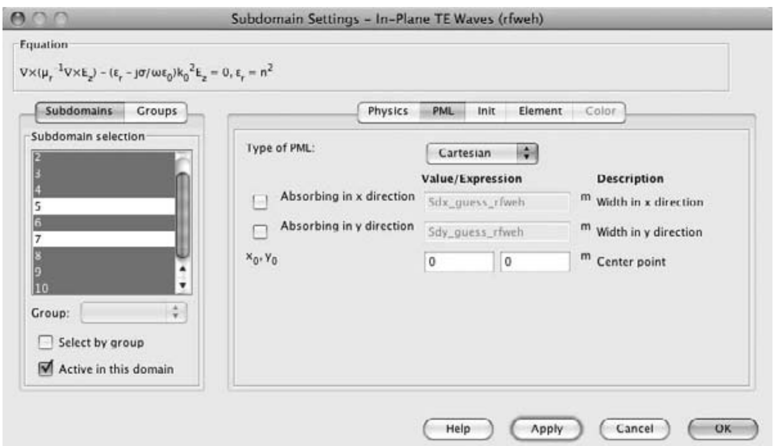


FIGURE 9.78 2D_PML_CM_1 model Subdomain Settings, PML type selection

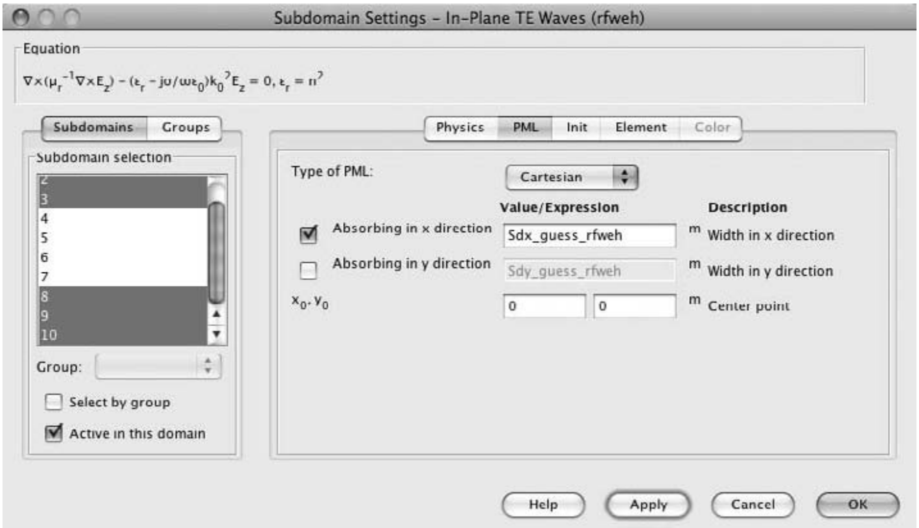


FIGURE 9.79 2D_PML_CM_1 model Subdomain Settings, *x* absorption

Select subdomains 1–3, and 8–10 (the vertical PMLs). Check the Absorbing in x direction check box. Click the Apply button. See Figure 9.79.

Select subdomains 1, 3, 4, 6, 8, and 10 (the horizontal PMLs). Check the Absorbing in y direction check box. Click the Apply button. See Figure 9.80.

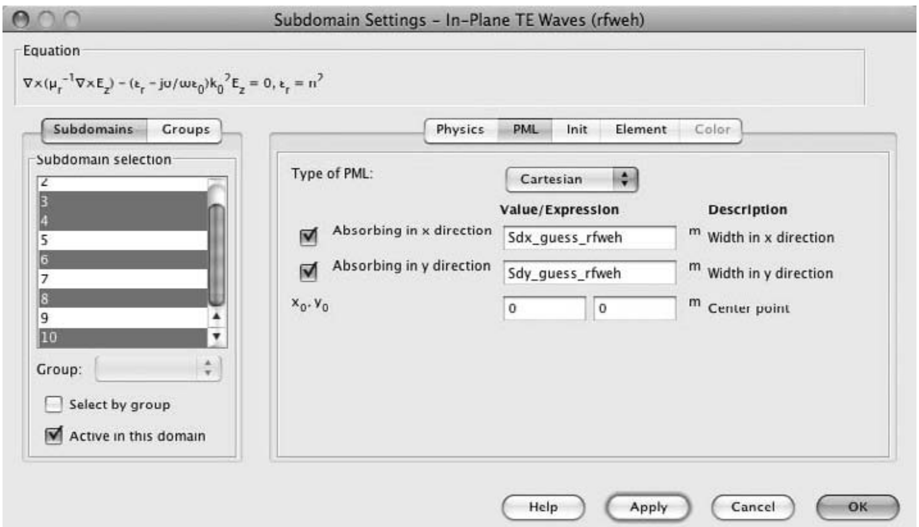


FIGURE 9.80 2D_PML_CM_1 model Subdomain Settings, *y* absorption

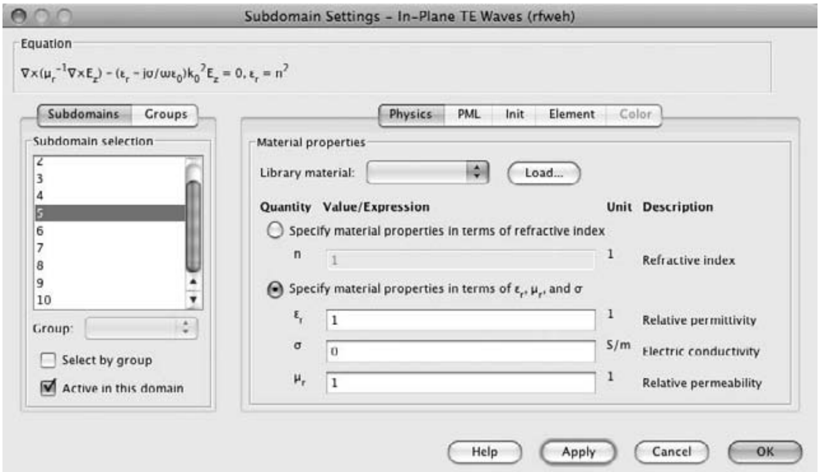


FIGURE 9.81 2D_PML_CM_1 model Subdomain Settings, Physics tab, subdomain 5

Click the Physics tab. Select subdomain 5 (the model domain). Enter $\varepsilon_r = 1$, $\sigma = 0$, and $\mu_r = 1$. Click the Apply button. See Figure 9.81.

Select subdomain 7 (the concave mirror). Click the Load button. Select Basic Material Properties > Copper. See Figure 9.82. Click OK.

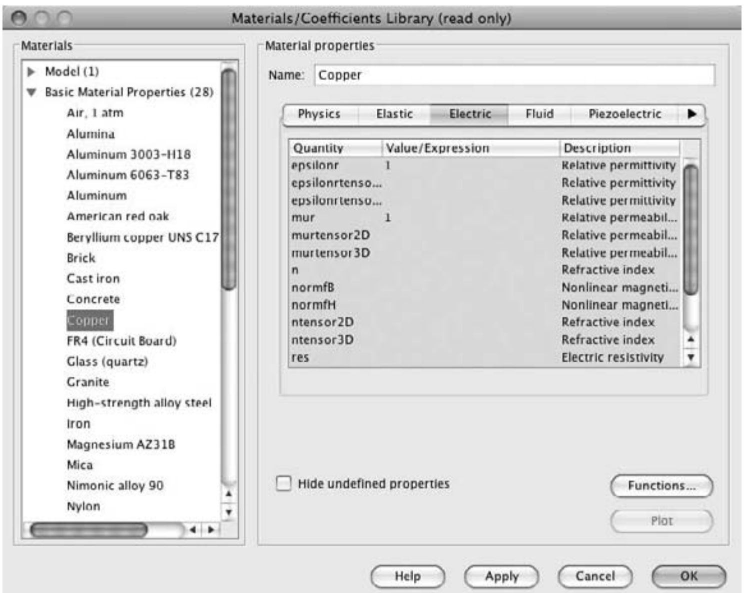


FIGURE 9.82 2D_PML_CM_1 model Materials/Coefficients Library, copper

FIGURE 9.83 2D_PML_CM_1 model Subdomain Settings, Physics tab, subdomain 7

See Figure 9.83. Click OK.

Physics Boundary Settings: In-Plane TE Waves (rfweh)

Having established the subdomain settings for the 2D_PML_CM_1 model, the next step is to define the fundamental Physics boundary settings. Using the menu bar, select Physics > Boundary Settings. Select boundary 1. Check the Select by Group check box to select the outer edge of the PMLs (boundaries). Select "Scattering boundary condition" from the Boundary condition pull-down list. See Figure 9.84. Click OK.

Mesh Generation

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 7 (the concave mirror). Enter 0.05 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 9.85.

Click the Mesh Selected button. Click the Select Remaining button. Click the Mesh Selected button. Click OK. See Figure 9.86.

Solving the 2D_PML_CM_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Parametric" in the Solver list. Enter lambda0_rfweh in the Parameter name edit window. Enter linspace(0.5,1.5,11) in the Parameter values edit window. (For later versions of the COMSOL Multiphysics software enter range(0.5,1/10,1.5) in the Parameter values edit window.) See Figure 9.87.

Click OK. Using the menu bar, select Solve > Solve Problem.

FIGURE 9.84 2D_PML_CM_1 model Boundary Settings

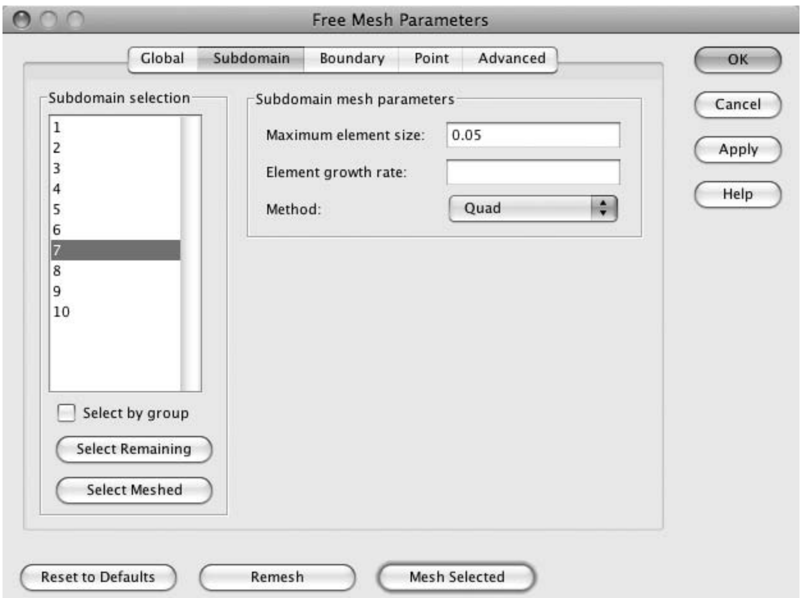


FIGURE 9.85 2D_PML_CM_1 model subdomain Free Mesh Parameters

FIGURE 9.86 2D_PML_CM_1 model mesh

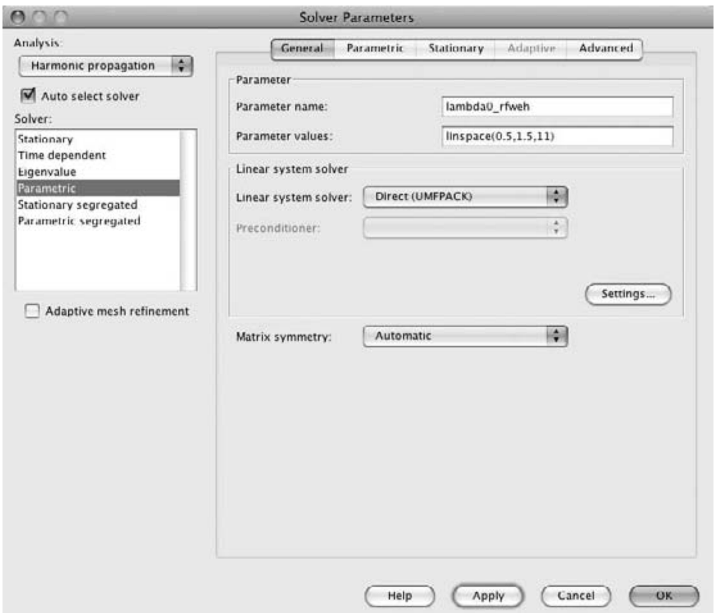


FIGURE 9.87 2D_PML_CM_1 model Solver Parameters

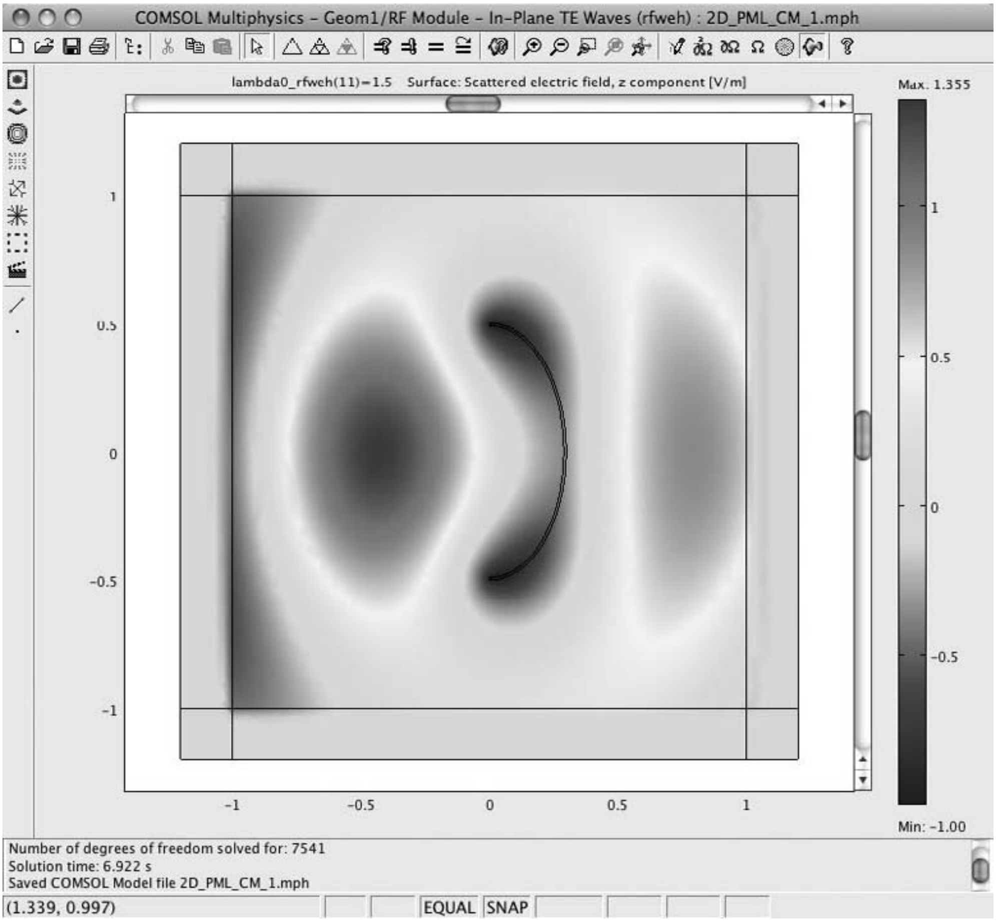


FIGURE 9.88 2D_PML_CM_1 model solution, scattered electric field, *z*-component (V/m)

Postprocessing and Visualization

The default plot shows a surface plot of the scattered electric field, *z*-component (V/m). See Figure 9.88.

An alternative approach to viewing the effect of the dielectric lens on the plane wave within the modeling domain is to suppress the plot within the PMLs and visualize the electric field, *z*-component. Using the menu bar, select Options > Suppress > Suppress Subdomains. Select subdomains 1–4, 6, and 8–10 (the PMLs). Click the Apply button. See Figure 9.89. Click OK.

Using the menu bar, select Postprocessing > Plot Parameters > Surface. Select “Electric field, z component” from the Predefined quantities pull-down list. See Figure 9.90.

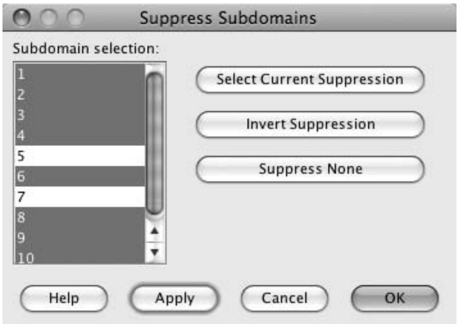


FIGURE 9.89 2D_PML_CM_1 model Suppress Subdomains

FIGURE 9.90 2D_PML_CM_1 model Plot Parameters, Surface tab

FIGURE 9.91 2D_PML_CM_1 model electric field, *z*-component

Click OK. See Figure 9.91.

Using the menu bar, select Postprocessing > Plot Parameters > Animate. Verify that all the Solutions to use are selected. See Figure 9.92. Click the Start Animation button.

2D Concave Mirror Model, with PMLs: Summary and Conclusions

The 2D concave mirror model, with PMLs (2D_PML_CM_1), has been built and solved. This model employs PMLs and a concave mirror to explore the geometric behavior of transverse electric field RF waves in the presence of a metallic focusing element (concave mirror). It can easily be observed by watching the animation that the position and intensity of the electric field, *z*-component varies greatly as a function of the free space wavelength.

FIGURE 9.92 2D_PML_CM_1 model Plot Parameters, Animate tab

2D Concave Mirror Model, without PMLs

The following numerical solution model (2D_NoPML_CM_1 model) is derived from the preceding concave mirror model (2D_PML_CM_1 model). In this case, however, the electromagnetic waves interact with a fixed, curved metallic mirror without PMLs at the boundaries of the modeling domain. The purpose of this model (2D_NoPML_CM_1) is to demonstrate empirically the difference between having or not having PMLs at the model boundaries.

To start building the 2D_NoPML_CM_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select RF Module > In-Plane Waves > TE Waves > Scattered harmonic propagation. See Figure 9.93. Click OK.

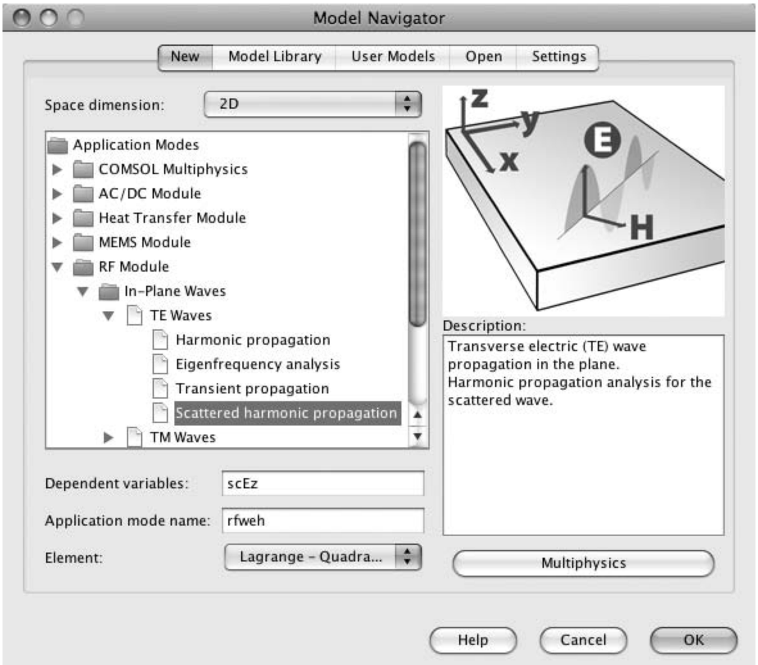


FIGURE 9.93 2D_NoPML_CM_1 Model Navigator setup

NOTE The Model Navigator command sequence (In-Plane Waves > TE Waves > Scattered harmonic propagation) selects a transverse electric field (z-direction) wave traveling in the plane (x, y-plane) of the modeling domain.

Select File > Save As. Enter 2D_NoPML_CM_1.mph in the Save As edit window. See Figure 9.94. Click the Save button.

Geometry Modeling

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes = 0.3, B-semiaxes = 0.5, Base = Center, x = 0, and y = 0. See Figure 9.95. Click OK.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes = 0.29, B-semiaxes = 0.49, Base = Center, x = 0, and y = 0. See Figure 9.96. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter Width = 0.5, Height = 1.0, Base = Corner, x = −0.5, and y = −0.5. See Figure 9.97. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter E1-E2-R1 in the Set formula edit window. See Figure 9.98.

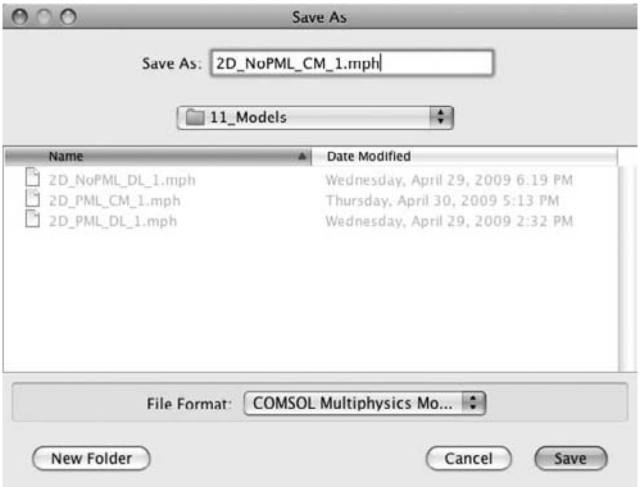


FIGURE 9.94 2D_NoPML_CM_1 model Save As edit window

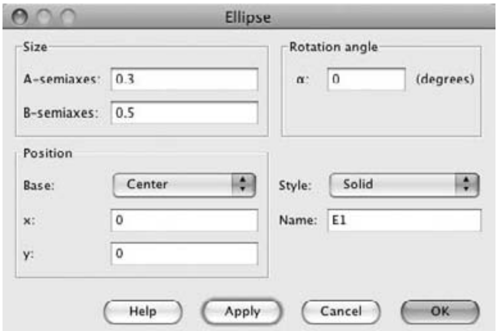


FIGURE 9.95 2D_NoPML_CM_1 model Ellipse (E1) edit window

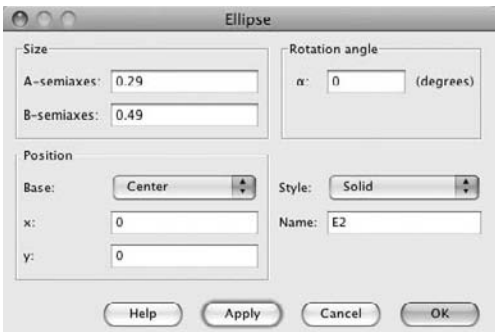


FIGURE 9.96 2D_NoPML_CM_1 model Ellipse (E2) edit window

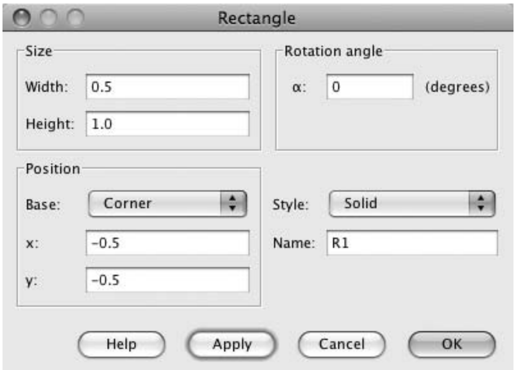


FIGURE 9.97 2D_NoPML_CM_1 model Rectangle (R1) edit window

Click OK. See Figure 9.99.

Using the menu bar, select Draw > Specify Objects > Square. Enter Width = 2.0, Base = Center, x = 0, and y = 0. See Figure 9.100.

Click OK, and then click the Zoom Extents button. See Figure 9.101.

Having established the geometry for the 2D_NoPML_CM_1 model, the next step is to define the fundamental Physics properties.

Physics Application Mode Properties: In-Plane TE Waves (rfweh)

Select Physics > Properties. Select "Free space wavelength" from the Specify wave using pull-down list. See Figure 9.102. Click OK.

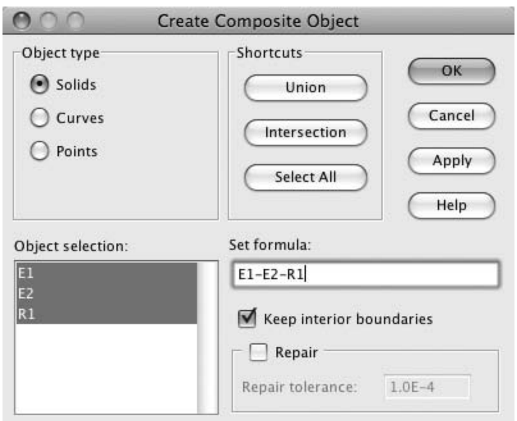


FIGURE 9.98 2D_NoPML_CM_1 model Create Composite Object edit window

FIGURE 9.99 2D_NoPML_CM_1 model concave mirror (CO1)

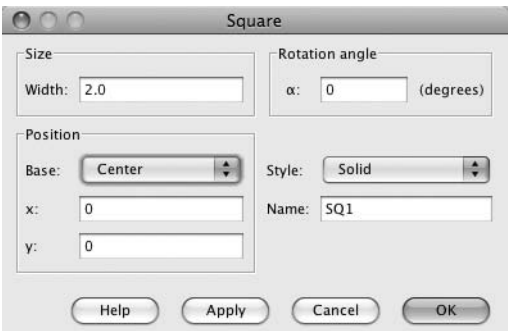


FIGURE 9.100 2D_NoPML_CM_1 model Square (SQ1) edit window

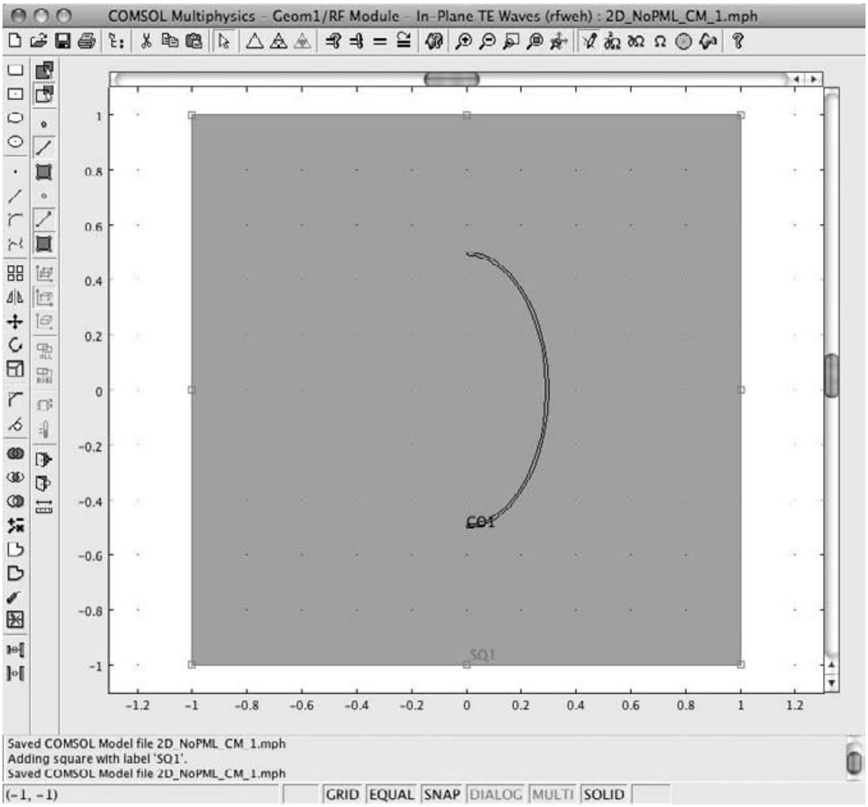


FIGURE 9.101 2D_NoPML_CM_1 model domain

FIGURE 9.102 2D_NoPML_CM_1 model Application Mode Properties edit window

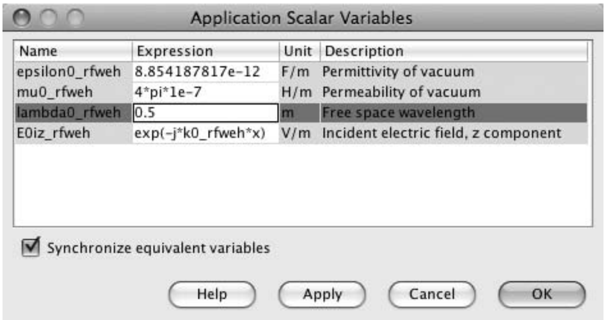


FIGURE 9.103 2D_NoPML_CM_1 model Application Scalar Variables (lambda0_rfweh) edit window

Physics Application Scalar Variables: In-Plane TE Waves (rfweh)

Select Physics > Scalar Variables. Enter 0.5 in the lambda0_rfweh edit window. See Figure 9.103. Click OK.

Physics Subdomain Settings: In-Plane TE Waves (rfweh)

Having established the basic Physics settings for the 2D_NoPML_CM_1 model, the next step is to define the fundamental Physics subdomain settings. Select Physics > Subdomain Settings. Click the Physics tab. Select subdomain 1 (the model domain). Enter $\varepsilon_r = 1$, $\sigma = 0$, and $\mu_r = 1$. Click the Apply button. See Figure 9.104.

FIGURE 9.104 2D_NoPML_CM_1 model Subdomain Settings, Physics tab, subdomain 1

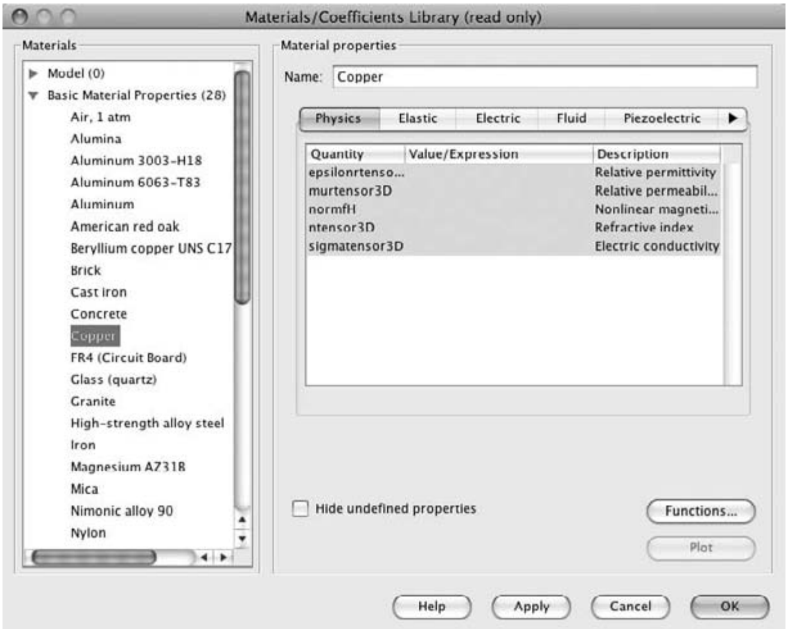


FIGURE 9.105 2D_NoPML_CM_1 model Materials/Coefficients Library, copper

Select subdomain 2 (the concave mirror). Click the Load button. Select Basic Material Properties > Copper. See Figure 9.105. Click OK.

See Figure 9.106. Click OK.

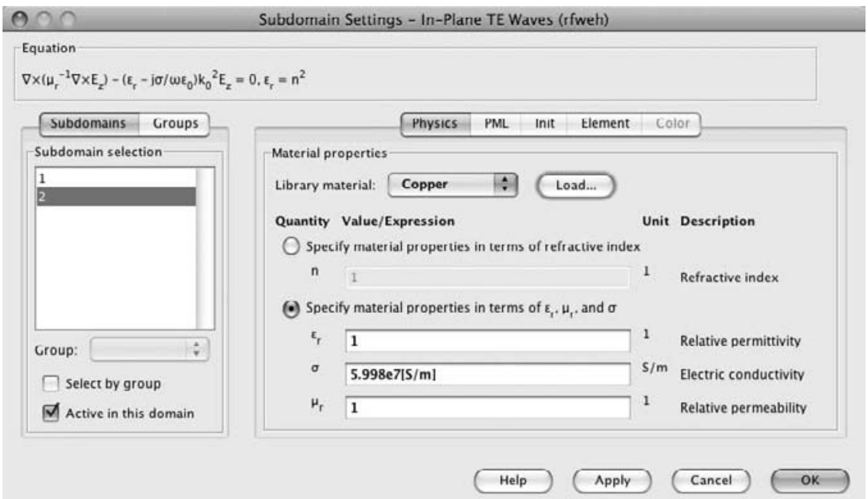


FIGURE 9.106 2D_NoPML_CM_1 model Subdomain Settings, Physics tab, subdomain 2

FIGURE 9.107 2D_NoPML_CM_1 model Boundary Settings

Physics Boundary Settings: In-Plane TE Waves (rfweh)

Having established the subdomain settings for the 2D_PML_CM_1 model, the next step is to define the fundamental Physics boundary settings. Using the menu bar, select Physics > Boundary Settings. Select boundaries 1, 2, 3, and 6 (the outer edges of the model domain). Select "Scattering boundary condition" from the Boundary condition pull-down list. See Figure 9.107. Click OK.

Mesh Generation

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 2 (the concave mirror). Enter 0.05 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 9.108.

Click the Mesh Selected button. Click the Select Remaining button. Click the Mesh Selected button. Click OK. See Figure 9.109.

Solving the 2D_NoPML_CM_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Parametric" in the Solver list. Enter lambda0_rfweh in the Parameter name edit window. Enter linspace(0.5,1.5,11) in the Parameter values edit window. (For later versions of the

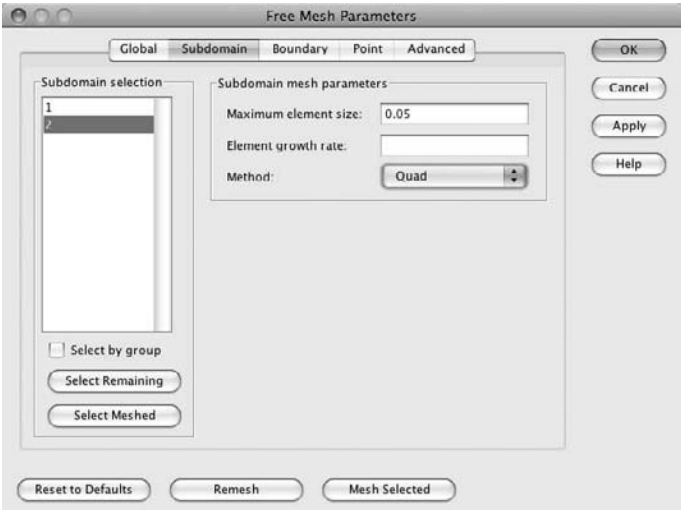


FIGURE 9.108 2D_NoPML_CM_1 model subdomain Free Mesh Parameters

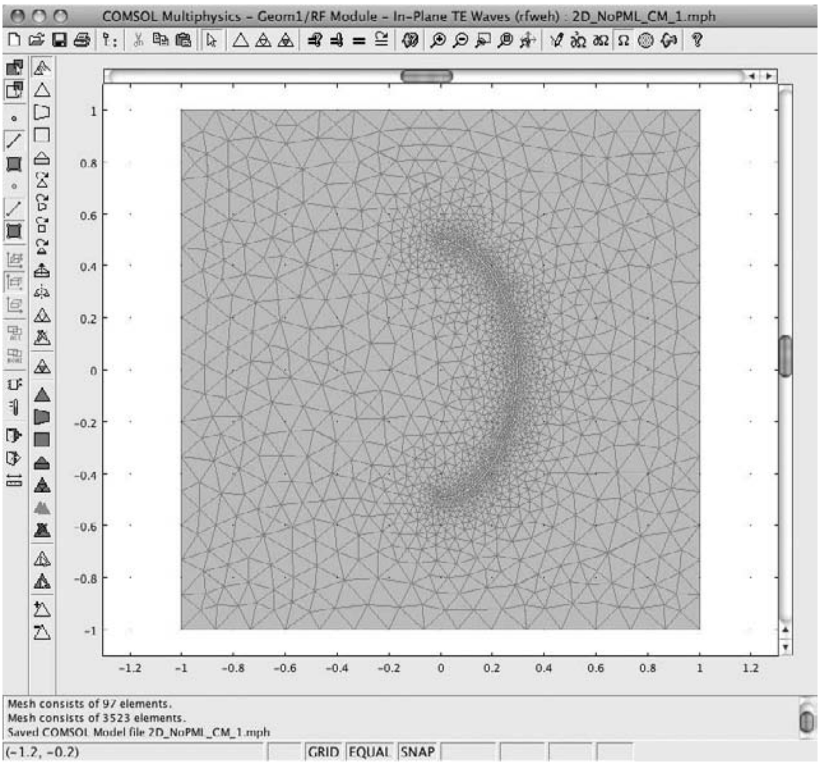


FIGURE 9.109 2D_NoPML_CM_1 model mesh

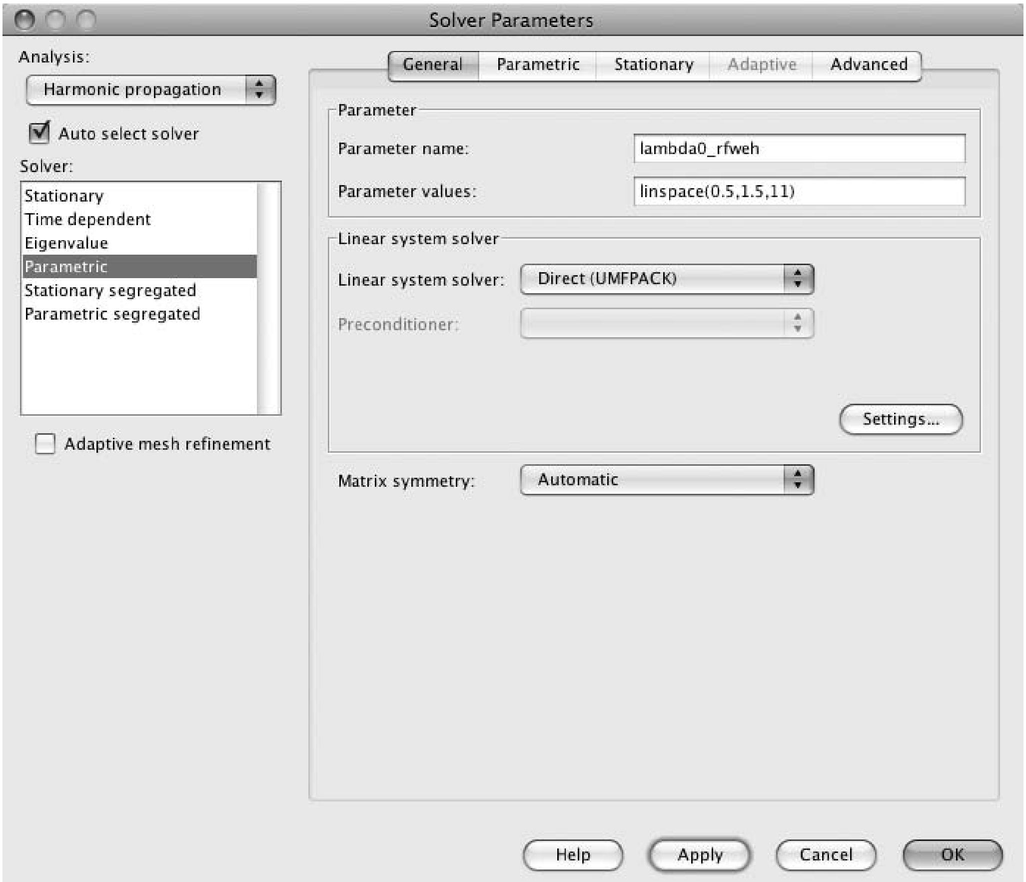


FIGURE 9.110 2D_NoPML_CM_1 model Solver Parameters

COMSOL Multiphysics software enter range(0.5,1/10,1.5) in the Parameter values edit window.) See Figure 9.110.

Click OK. Using the menu bar, select Solve > Solve Problem.

Postprocessing and Visualization

The default plot shows a surface plot of the scattered electric field, *z*-component (V/m). See Figure 9.111.

An alternative approach to viewing the effect of the dielectric lens on the plane wave within the modeling domain is to visualize the electric field, *z*-component.

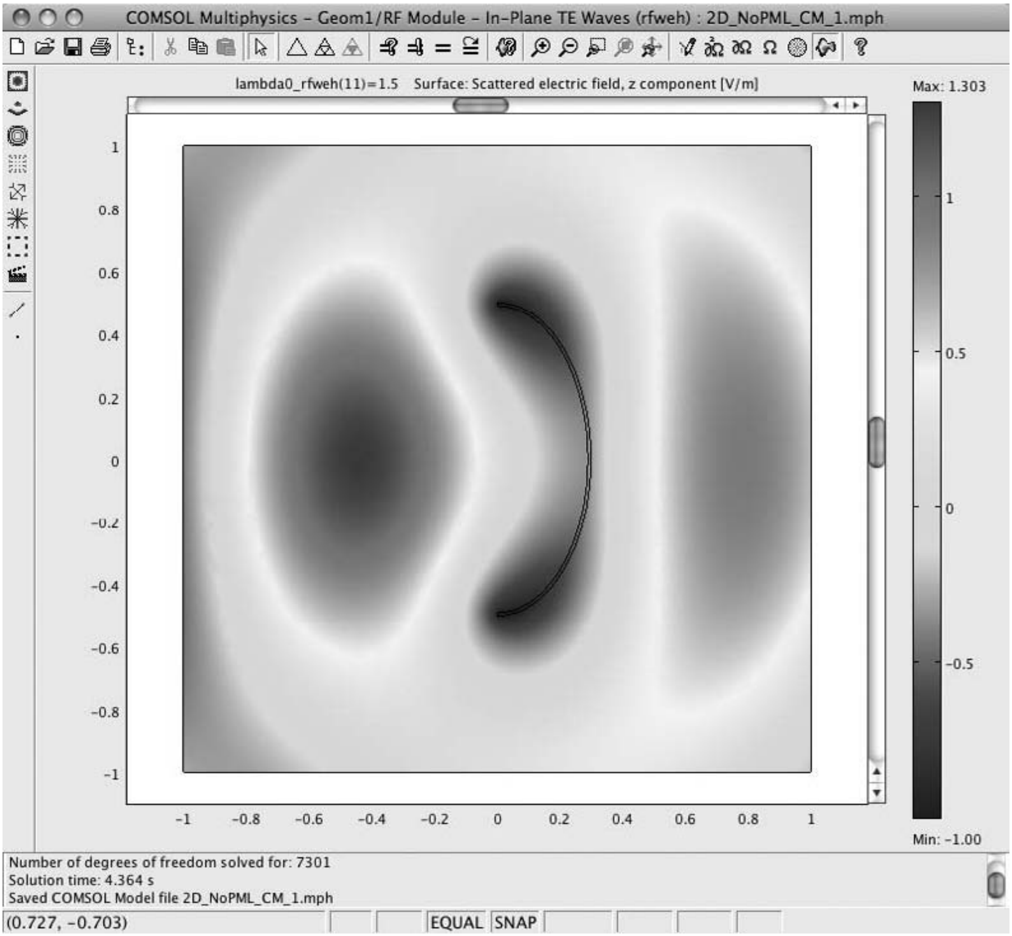


FIGURE 9.111 2D_NoPML_CM_1 model solution, scattered electric field, *z*-component (V/m)

Using the menu bar, select Postprocessing > Plot Parameters > Surface. Select "Electric field, z component" from the Predefined quantities pull-down list. See Figure 9.112.

Click OK. See Figure 9.113.

Using the menu bar, select Postprocessing > Plot Parameters > Animate. Verify that all the Solutions to use are selected. See Figure 9.114. Click the Start Animation button.

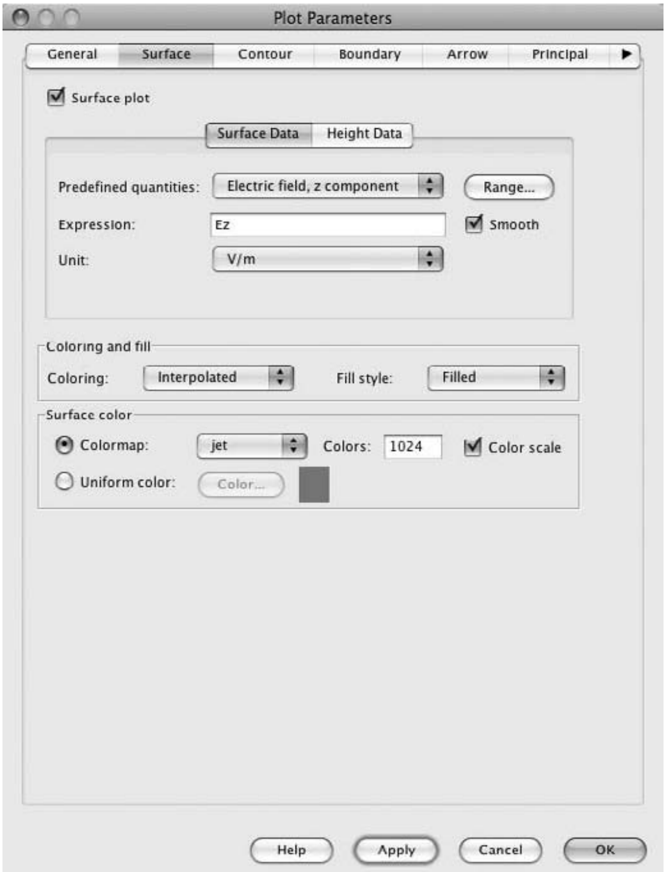


FIGURE 9.112 2D_NoPML_CM_1 model Plot Parameters, Surface tab

2D Concave Mirror Model, with and without PMLs: Summary and Conclusions

The 2D concave mirror models, with and without PMLs (2D_PML_CM_1 and 2D_NoPML_CM_1, respectively), have been built and solved. The best method of comparison between the two models is to view visualizations for the electric field, *z*-component for the same wavelength from each model together. Figures 9.115 through 9.120 show visualizations for 0.5 m (Figures 9.115 and 9.116), 1.0 m (Figures 9.117 and 9.118), and 1.5 m (Figures 9.119 and 9.120).

In comparison to the dielectric lens models presented in the first half of this chapter, it is apparent that there are also only small differences in the electric field, *z*-component visualizations between the PML and no-PML models for the concave mirror. This lack of large differences between the PML and no-PML models shows

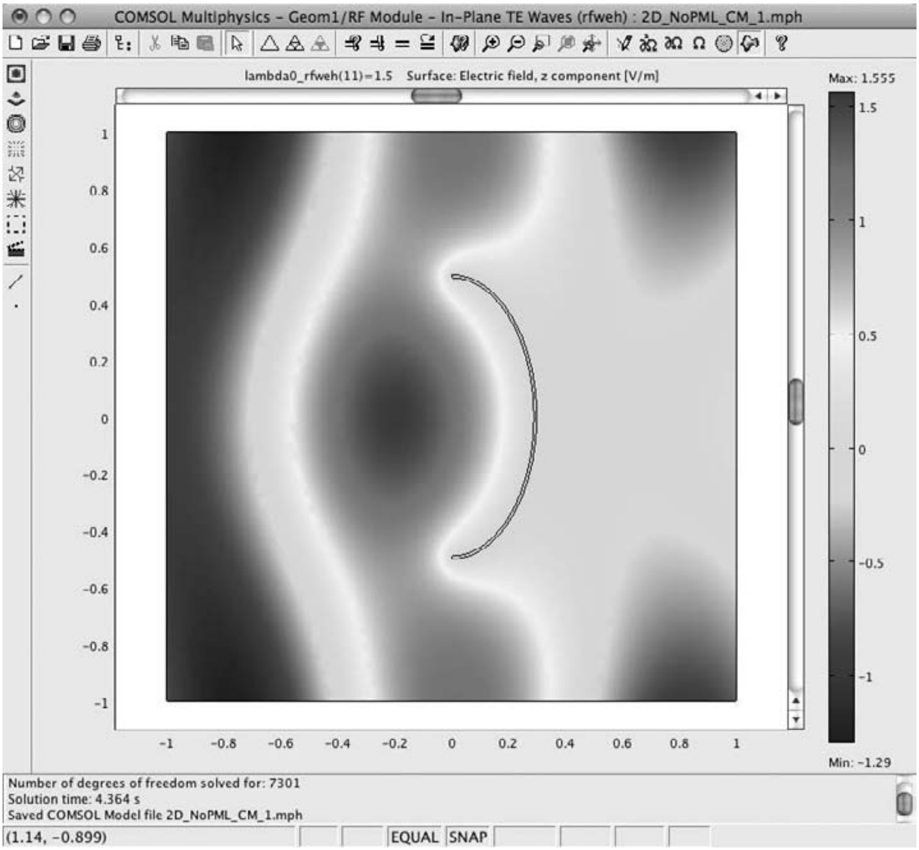


FIGURE 9.113 2D_NoPML_CM_1 model electric field, *z*-component

the modeler that he or she needs to understand the relative importance of the modeled values needed to evaluate the application and the application environment so as to build the best model. The PML model best approximates a free space environment (no reflections). For other environments, the modeler needs to determine the best boundary condition approximation using standard practices and a first principles approach. When all else fails (or even before), do a first principles analysis of the environment before building the model.

NOTE Why do the solutions of the two dielectric lens models differ significantly and the solutions of the two concave mirror models converge to similar solutions? Consider the fact that a lossless dielectric is electromagnetically transparent and a metal mirror (e.g., copper) is electromagnetically opaque. Then a first principles analysis should answer the question.

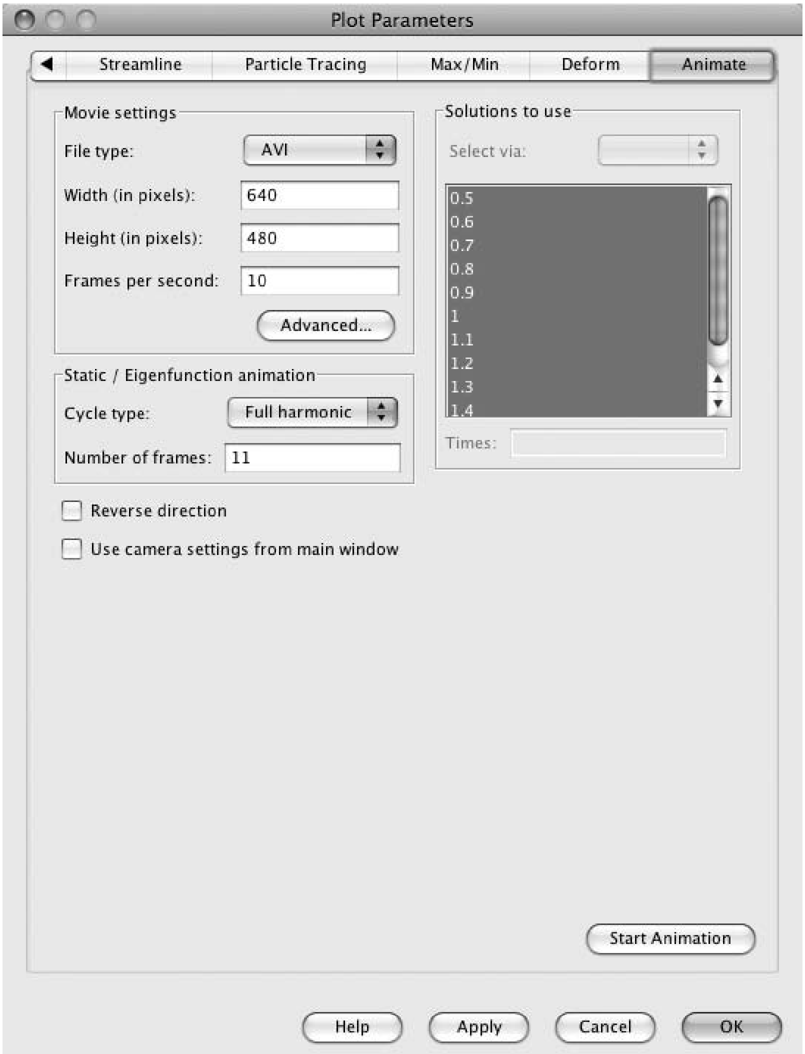


FIGURE 9.114 2D_NoPML_CM_1 model Plot Parameters, Animate tab

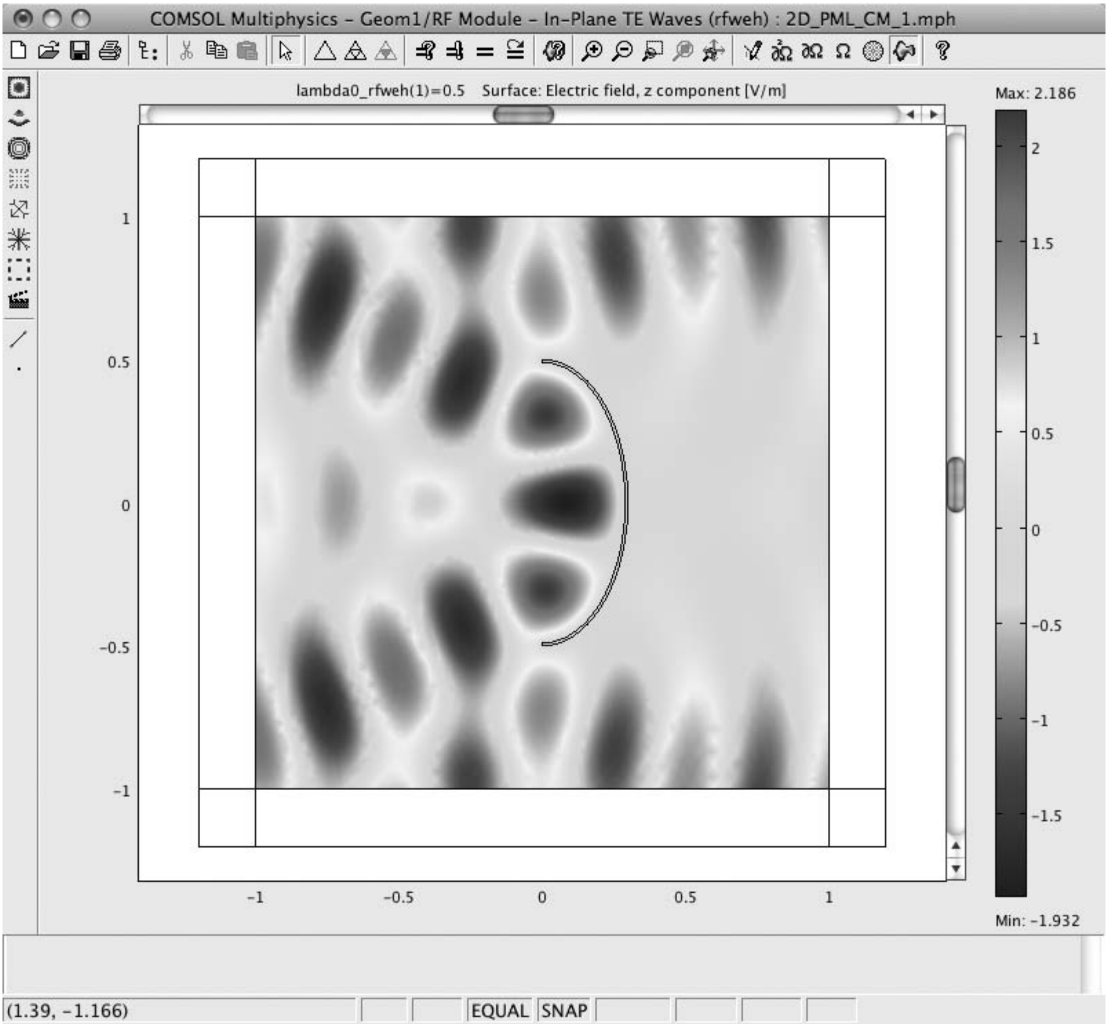


FIGURE 9.115 2D_PML_CM_1 model plot electric field, *z*-component, 0.5 m

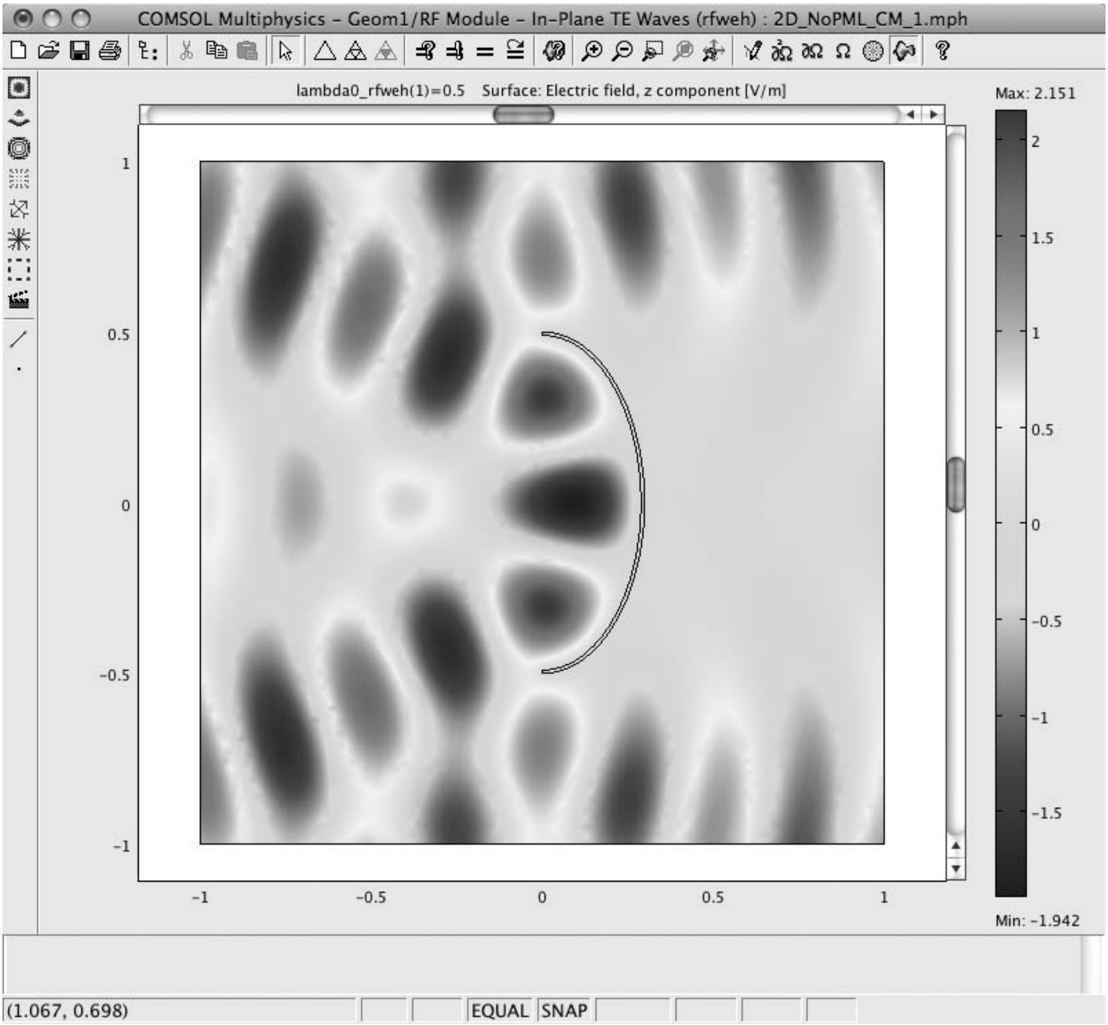


FIGURE 9.116 2D_NoPML_CM_1 model plot electric field, *z*-component, 0.5 m

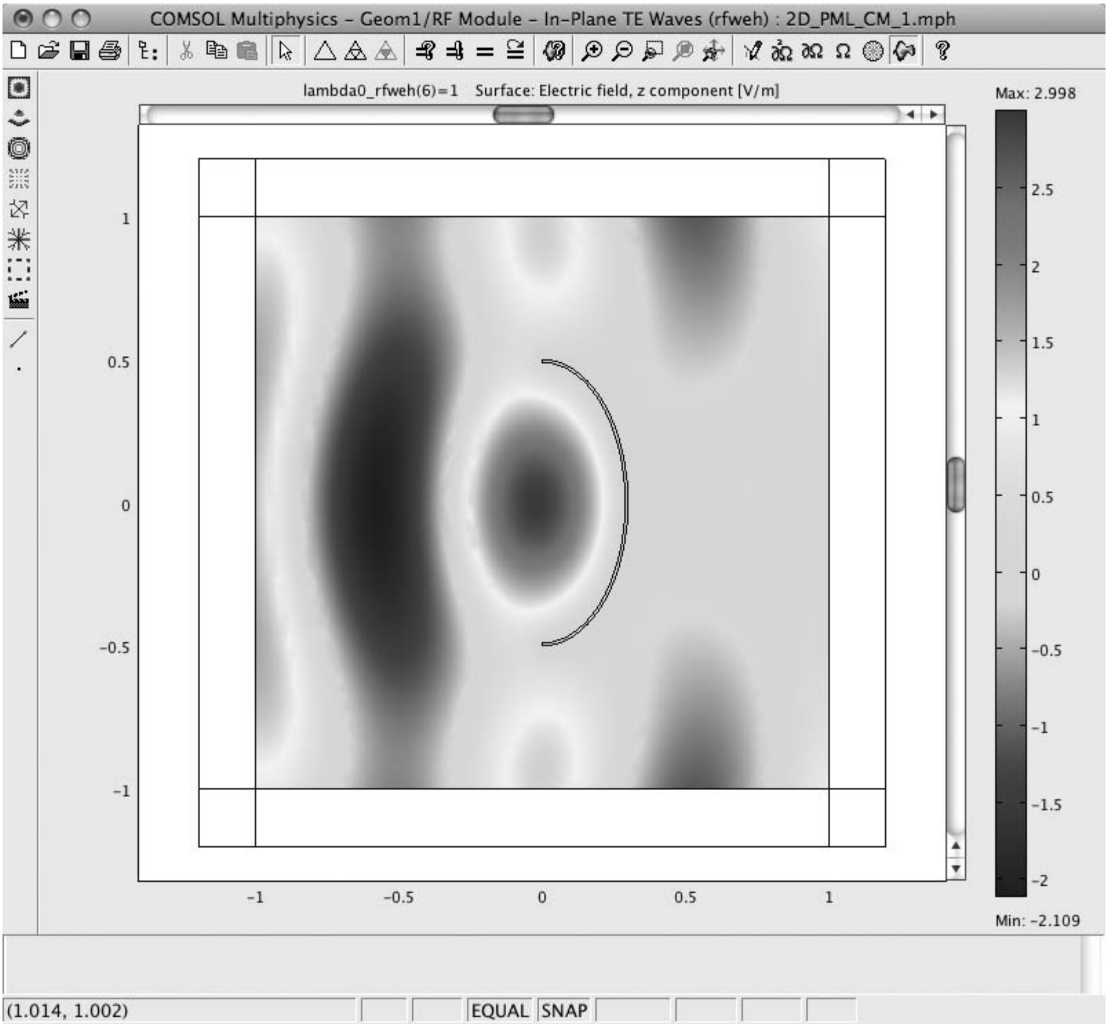


FIGURE 9.117 2D_PML_CM_1 model plot electric field, *z*-component, 1.0 m

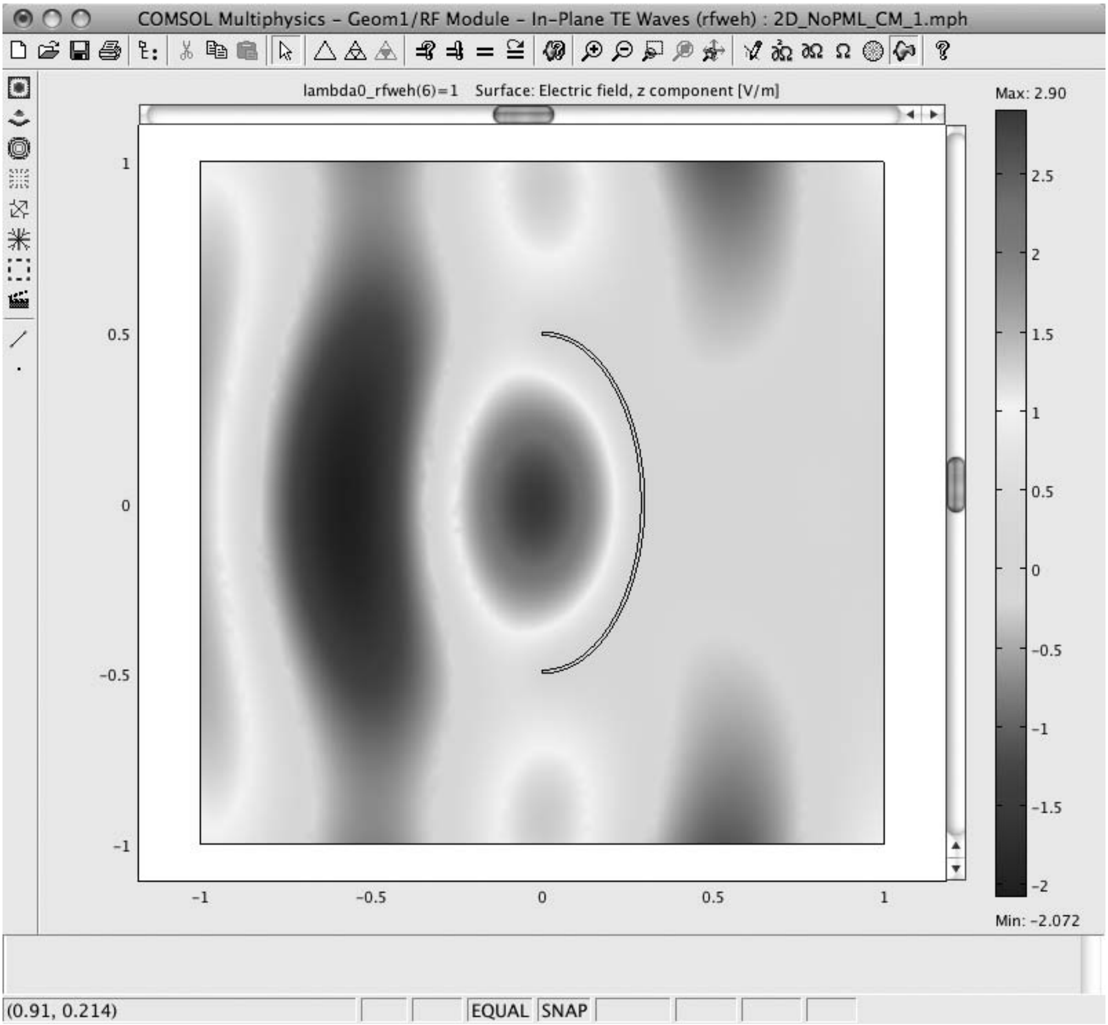


FIGURE 9.118 2D_NoPML_CM_1 model plot electric field, *z*-component, 1.0 m

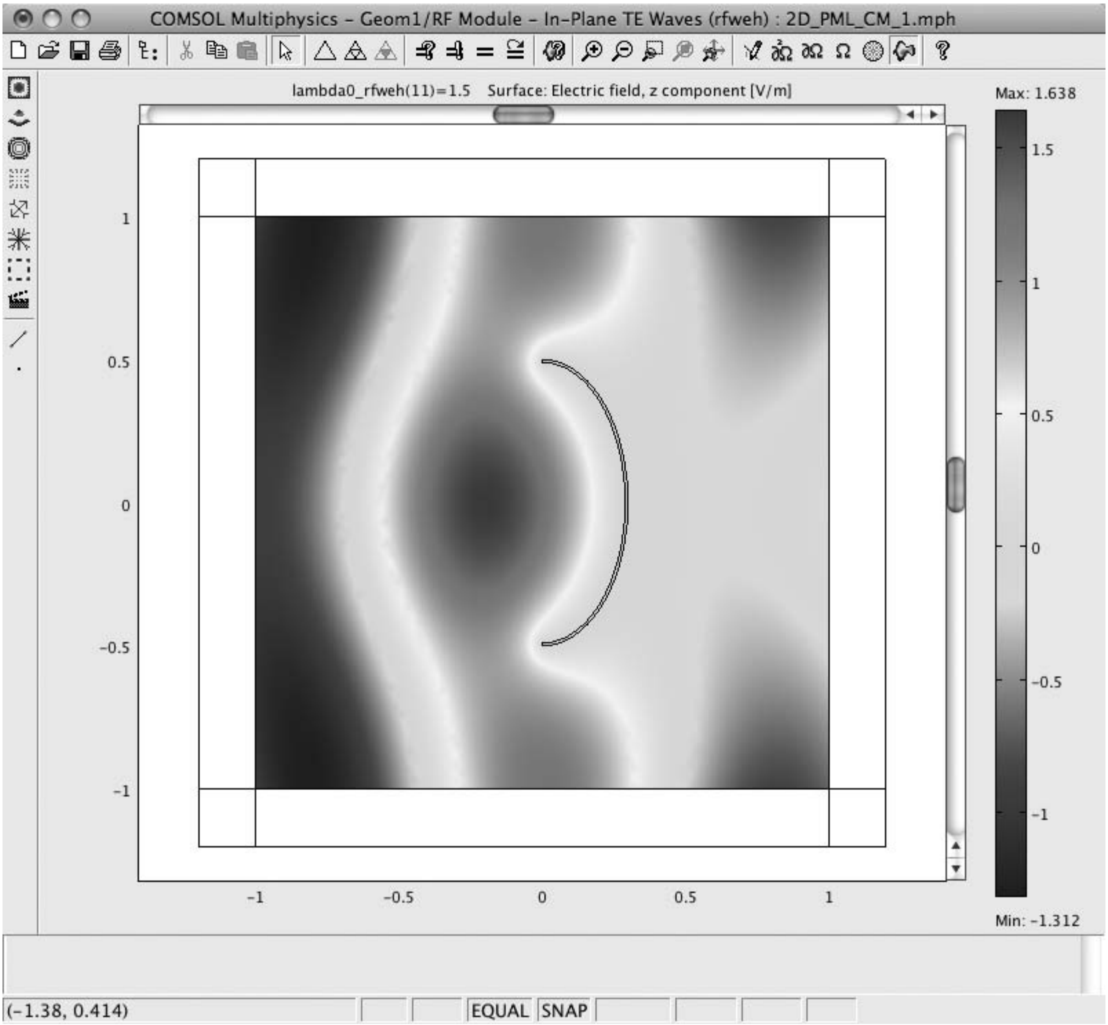


FIGURE 9.119 2D_PML_CM_1 model plot electric field, *z*-component, 1.5 m

FIGURE 9.120 2D_NoPML_CM_1 model plot electric field, *z*-component, 1.5 m

References

1. http://en.wikipedia.org/wiki/Maxwell%27s_Equations
2. http://en.wikipedia.org/wiki/Boundary_conditions
3. J. P. Berenger, “A perfectly matched layer for the absorption of electromagnetic waves”, J. Comput. Phys., Vol. 114, No. 2, 1994, pp. 185–200.
4. http://en.wikipedia.org/wiki/Finite-difference_time-domain_method
5. http://en.wikipedia.org/wiki/Perfectly_matched_layer
6. http://math.mit.edu/~stevenj/18.369/pml.pdf

7. *COMSOL RF Module User's Guide,* Version 3.4, October 2007, COMSOL AB, Stockholm, Sweden, pp. 43–48.

8. http://en.wikipedia.org/wiki/Concave_mirror

Exercises

1. Build, mesh, and solve the 2D dielectric lens model, with PMLs, problem presented in this chapter.
2. Build, mesh, and solve the 2D dielectric lens model, without PMLs, problem presented in this chapter.
3. Build, mesh, and solve the 2D concave mirror model, with PMLs, problem presented in this chapter.
4. Build, mesh, and solve the 2D concave mirror model, without PMLs, problem presented in this chapter.
5. Explore other materials as applied in the 2D dielectric lens model, with PMLs.
6. Explore other materials as applied in the 2D dielectric lens model, without PMLs.
7. Explore other materials as applied in the 2D concave mirror model, with PMLs.
8. Explore other materials as applied in the 2D concave mirror model, without PMLs.
9. Explore the different geometries in the 2D dielectric lens model, with PMLs.
10. Explore the different geometries in the 2D concave mirror model, with PMLs.

10 Bioheat Models

In This Chapter

■ Bioheat Modeling Guidelines and Coordinate Considerations

Bioheat Equation Theory

For the new modeler or those readers unfamiliar with this topic, bioheat modeling is the development of models for the analysis of heat transfer in materials (e.g., tissues, fluids) and systems derived from or related to currently or previously living organisms. The solution of the bioheat equation as applied to particular models is most important, obviously, when those models are developed to explore potential techniques for critical therapeutic applications (e.g., destroying cancer cells, killing tumors).

In August 1948, Harry H. Pennes published his landmark paper "Analysis of Tissue and Arterial Blood Temperatures in the Resting Human Forearm."[1] In that paper, he proposed that heat flow is proportional to the difference in temperature between the arterial blood and the local tissue. Pennes's work is considered fundamental in this area of study and has since been cited extensively.[2]

In the COMSOL® Multiphysics® software, the bioheat equation (Pennes equation) takes the form of an application mode within the Heat Transfer Module. In the Bioheat Equation Application Mode, the bioheat equation is formulated as follows:

$$\delta_{ts}\rho C\frac{\partial T}{\partial t} + \nabla \cdot (-\vec{k}\nabla T) = \rho_b C_b \omega_b (T_b - T) + Q_{met} + Q_{ext} \qquad (10.1)$$

where δ_{ts} = time-scaling coefficient (default value = 1; dimensionless)
ρ = tissue density (kg/m^3)
C = tissue heat capacity [J/(kg · K)]
T = temperature (K)
$\overrightarrow{k}$ = tissue thermal conductivity tensor [W/(m · K)]
ρ_b = blood density (kg/m^3)
C_b = blood heat capacity [J/(kg·K)]
ω_b = blood perfusion rate [m^3/(m^3·s)]
T_b = temperature, arterial blood (K)
Q_{met} = metabolic heat source (W/m^3)
Q_{ext} = external environmental heat source (W/m^3)

NOTE The perfusion[3] rate is the rate at which a fluid (e.g., blood) flows through a type of tissue (e.g., muscle, heart, liver). It is, of course, very important to know the correct perfusion value for the tissue/fluid type in question.

Even though equation 10.1 is shown as formulated for blood flow, it can be equally well employed for other fluids or fluid compositions under the appropriate circumstances (e.g., artificial blood, different animal-life fluids). When employing variations of the formulation of the bioheat equation, modelers need to carefully verify the underlying assumptions employed in their particular model.

The bioheat equation is similar to the conduction heat equation. In the case of steady-state heat flow, the first term on the left vanishes:

$$\delta_{ts}\rho C\frac{\partial T}{\partial t} = 0 \tag{10.2}$$

In the bioheat equation, what would normally be the single heat source term on the left side of the heat conduction equation (Q) is now separated into three terms.

The perfusion term: $\rho_b C_b \omega_b (T_b - T)$ (10.3)

The metabolic term: Q_{met} (10.4)

The external source term: Q_{ext} (10.5)

NOTE The division of the normally single heat source term in the bioheat equation into three terms is done to facilitate a conceptual linkage and to ease the formulation of the PDE when creating models for this type of problem (biological).

The bioheat equation, as constructed by Pennes, constitutes a good first-order approximation to the physical processes (thermal conduction) involved in the solution

of the heat transfer problem for biological specimens. This formulation is typically adequate for the modeling of most biological problems. More terms can, of course, be added if perceived as necessary, albeit at the risk of increased complexity, associated model size, and computational time.

However, because the bioheat equation already serves the needed level of accuracy for a typical decision point, little additional beneficial knowledge will be gained from the addition of second-order effects to the equation, considering the intrinsic fundamental limits of most biological system model problems.

Tumor Laser Irradiation Theory

The optical coefficient of absorption for laser photons (irradiation) of tumors does not generally differ significantly from the optical coefficient of absorption for the surrounding tissue. To develop this laser irradiation therapeutic methodology, it is necessary to raise the local absorption coefficient by artificial means. The change in absorption coefficient is accomplished by injection into the tumor of a designed high-absorption material.[4] This type of procedure is usually designated as a minimally invasive procedure.

The laser beam energy contributes a heat source to the bioheat equation as follows:

$$Q_{\text{laser}} = I_0 a e^{az - \frac{r^2}{2\sigma^2}} \tag{10.6}$$

where I_0 = irradiation intensity (W/m^2)
a = absorptivity (1/m)
σ = irradiated region width parameter (m)

2D Axisymmetric Tumor Laser Irradiation Model

The following numerical solution model (2D_Bio_TLI_1 model) is derived from a model that was originally developed by COMSOL as a Heat Transfer Module tutorial model for the demonstration of the solution of a bioheat equation model. That model was developed for distribution with the Heat Transfer Module software as part of the COMSOL Heat Transfer Module Model Library.

NOTE The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. The guiding principle needs to be that tumor cells die at elevated temperatures. The literature cites temperatures that range from 42 °C (315.15 K)[5] to 60 °C (333.15 K).[6] If the postulated method raises the local temperature of the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful.

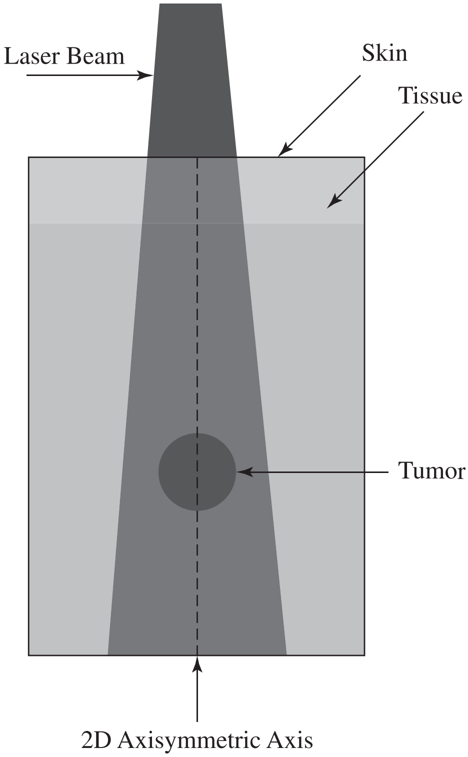


FIGURE 10.1 2D_Bio_TLI_1 model modeling domain overview

This first model takes advantage of the transparency of human tissue in certain infrared (IR) wavelengths.[7] Figure 10.1 shows the structure of the modeling domain. Because the model is created as a 2D axisymmetric model, only the right half of the structure will be used in the calculations.

To start building the 2D_Bio_TLI_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > Bioheat Equation > Transient analysis. See Figure 10.2. Click OK.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 10.1; see also Figure 10.3. Click OK.

Geometry Modeling

Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 10.2. Click OK after filling in the parameters of each separate rectangle in the Rectangle edit window. See Figures 10.4 and 10.5.

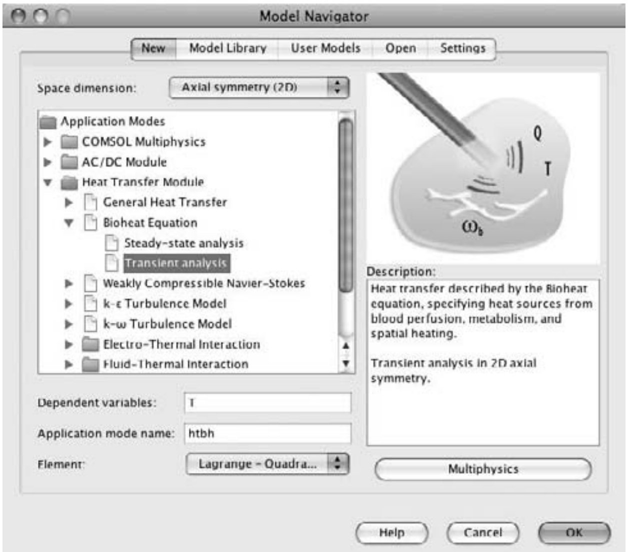
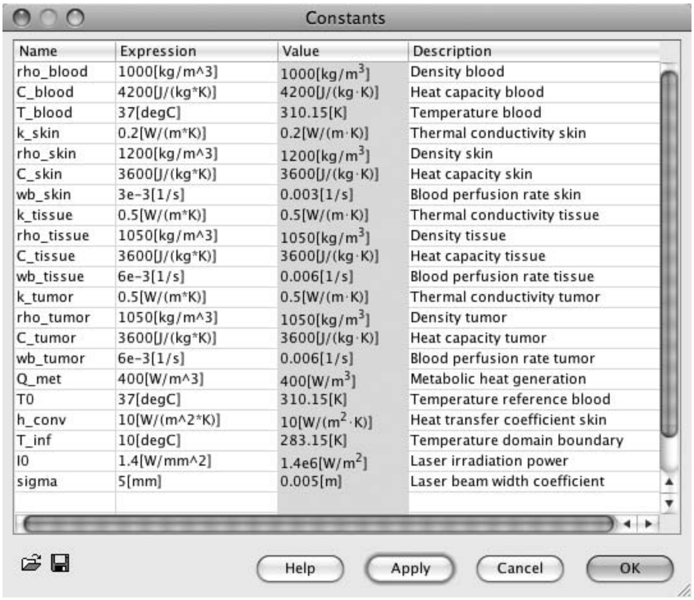


FIGURE 10.2 2D_Bio_TLI_1 Model Navigator setup

Table 10.1 Constants Edit Window

Name	Expression	Description
rho_blood	1000[kg/m^3]	Density blood
C_blood	4200[J/(kg*K)]	Heat capacity blood
T_blood	37[degC]	Temperature blood
k_skin	0.2[W/(m*K)]	Thermal conductivity skin
rho_skin	1200[kg/m^3]	Density skin
C_skin	3600[J/(kg*K)]	Heat capacity skin
wb_skin	3e-3[1/s]	Blood perfusion rate skin
k_tissue	0.5[W/(m*K)]	Thermal conductivity tissue
rho_tissue	1050[kg/m^3]	Density tissue
C_tissue	3600[J/(kg*K)]	Heat capacity tissue
wb_tissue	6e-3[1/s]	Blood perfusion rate tissue
k_tumor	0.5[W/(m*K)]	Thermal conductivity tumor
rho_tumor	1050[kg/m^3]	Density tumor
C_tumor	3600[J/(kg*K)]	Heat capacity tumor
wb_tumor	6e-3[1/s]	Blood perfusion rate tumor
Q_met	400[W/m^3]	Metabolic heat generation
T0	37[degC]	Temperature reference blood
h_conv	10[W/(m^2*K)]	Heat transfer coefficient skin
T_inf	10[degC]	Temperature domain boundary
I0	1.4[W/mm^2]	Laser irradiation power
sigma	5[mm]	Laser beam width coefficient



Name	Expression	Value	Description
rho_blood	1000[kg/m^3]	1000[kg/m^3]	Density blood
C_blood	4200[J/(kg*K)]	4200[J/(kg·K)]	Heat capacity blood
T_blood	37[degC]	310.15[K]	Temperature blood
k_skin	0.2[W/(m*K)]	0.2[W/(m·K)]	Thermal conductivity skin
rho_skin	1200[kg/m^3]	1200[kg/m^3]	Density skin
C_skin	3600[J/(kg*K)]	3600[J/(kg·K)]	Heat capacity skin
wb_skin	3e-3[1/s]	0.003[1/s]	Blood perfusion rate skin
k_tissue	0.5[W/(m*K)]	0.5[W/(m·K)]	Thermal conductivity tissue
rho_tissue	1050[kg/m^3]	1050[kg/m^3]	Density tissue
C_tissue	3600[J/(kg*K)]	3600[J/(kg·K)]	Heat capacity tissue
wb_tissue	6e-3[1/s]	0.006[1/s]	Blood perfusion rate tissue
k_tumor	0.5[W/(m*K)]	0.5[W/(m·K)]	Thermal conductivity tumor
rho_tumor	1050[kg/m^3]	1050[kg/m^3]	Density tumor
C_tumor	3600[J/(kg*K)]	3600[J/(kg·K)]	Heat capacity tumor
wb_tumor	6e-3[1/s]	0.006[1/s]	Blood perfusion rate tumor
Q_met	400[W/m^3]	400[W/m^3]	Metabolic heat generation
T0	37[degC]	310.15[K]	Temperature reference blood
h_conv	10[W/(m^2*K)]	10[W/(m^2·K)]	Heat transfer coefficient skin
T_inf	10[degC]	283.15[K]	Temperature domain boundary
I0	1.4[W/mm^2]	1.4e6[W/m^2]	Laser irradiation power
sigma	5[mm]	0.005[m]	Laser beam width coefficient

FIGURE 10.3 2D_Bio_TLI_1 model Constants (R1) edit window

Table 10.2 Geometry Components

Name	Width	Height	Base	r	z	Figure Number
R1	0.1	0.09	Corner	−0.05	−0.1	10.4
R2	0.1	0.01	Corner	−0.05	−0.01	10.5

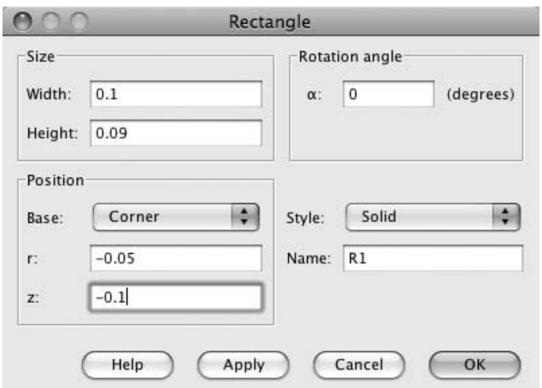


FIGURE 10.4 2D_Bio_TLI_1 model Rectangle (R1) edit window

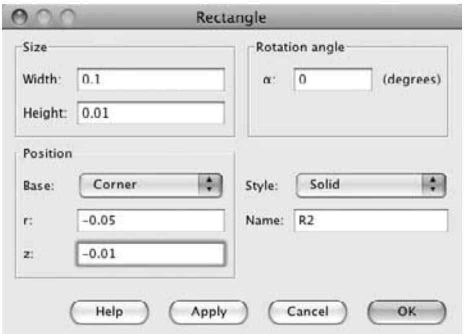


FIGURE 10.5 2D_Bio_TLI_1 model Rectangle (R2) edit window

Click the Zoom Extents button. See Figure 10.6.

Select File > Save As. Enter 2D_Bio_TLI_1.mph in the Save As edit window. See Figure 10.7. Click the Save button.

Using the menu bar, select Draw > Specify Objects > Circle. Enter Radius = 0.005, Base = Center, r = 0, and z = −0.05. See Figure 10.8. Click OK.

FIGURE 10.6 2D_Bio_TLI_1 model rectangles (R1, R2)

FIGURE 10.7 2D_Bio_TLI_1 model Save As edit window

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter Width = 0.05, Height = 0.1, Base = Corner, r = −0.05, and z = −0.1. See Figure 10.9. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter R1+R2+C1−R3 in the Set formula edit window. Check the Keep interior boundaries check box. See Figure 10.10.

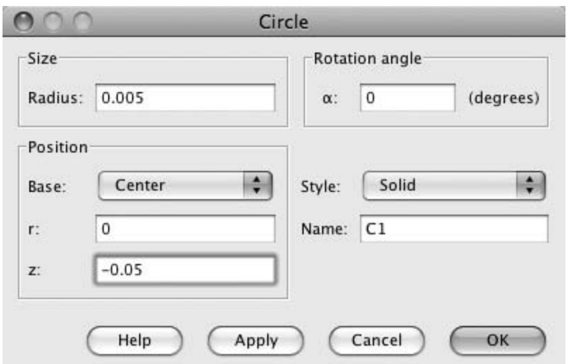


FIGURE 10.8 2D_Bio_TLI_1 model Circle (C1) edit window

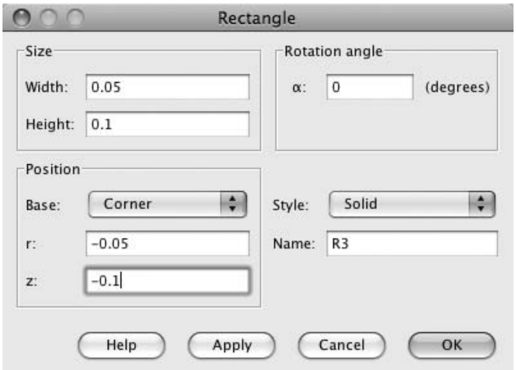


FIGURE 10.9 2D_Bio_TLI_1 model Rectangle (R3) edit window

Click OK. See Figure 10.11.

Having established the geometry for the 2D_Bio_TLI_1 model, the next step is to define the fundamental Physics properties.

Physics Settings: Scalar Expressions

Select Options > Expressions > Scalar Expressions. Enter Name = Q_laser. Enter Expression = I0*a*exp(a*z-r^2/(2*sigma^2)). See Figure 10.12. Click OK.

Physics Settings: Subdomain Expressions

Select Options > Expressions > Subdomain Expressions.

NOTE In the entries in the Subdomain Expressions window, the variable Name a needs to be entered only once, as indicated by the following instructions.

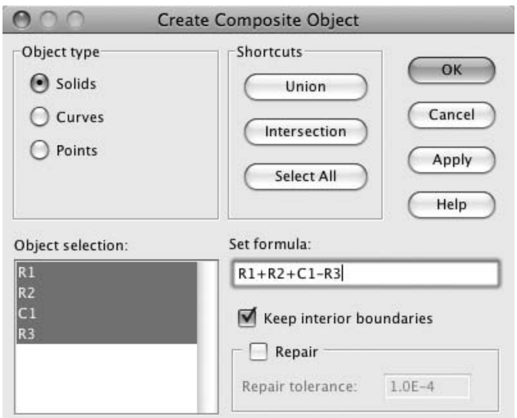


FIGURE 10.10 2D_Bio_TLI_1 model Create Composite Object edit window

FIGURE 10.11 2D_Bio_TLI_1 model domain: skin, tissue, and tumor (CO1)

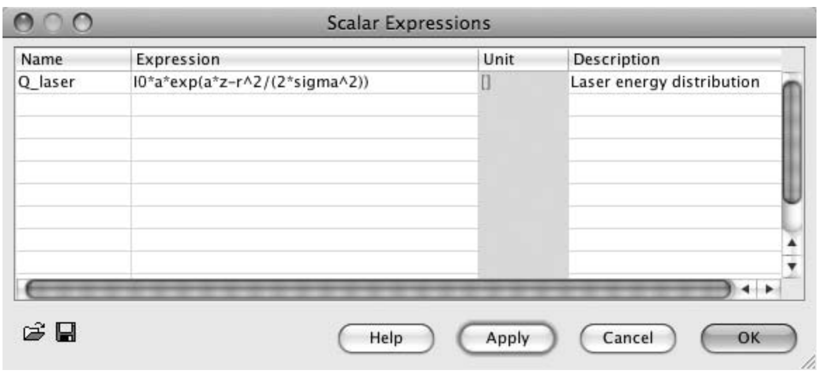


FIGURE 10.12 2D_Bio_TLI_1 model Scalar Expressions edit window

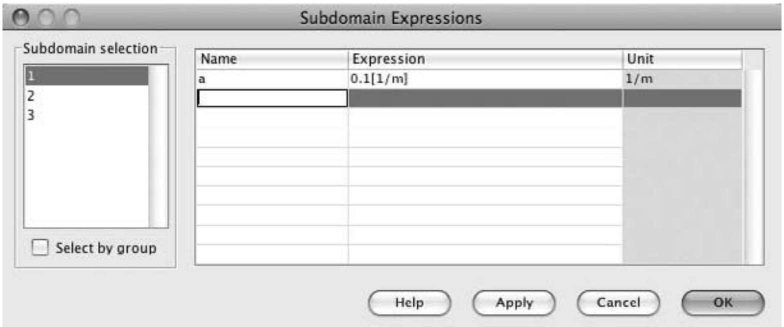


FIGURE 10.13 2D_Bio_TLI_1 model Subdomain Expressions (1) edit window

For subdomain 1, enter Name = a, Expression = 0.1[1/m]. See Figure 10.13.
For subdomain 2, enter Expression = 4[1/m]. See Figure 10.14.
For subdomain 3, enter Expression = 0.1[1/m]. See Figure 10.15.
Click OK.

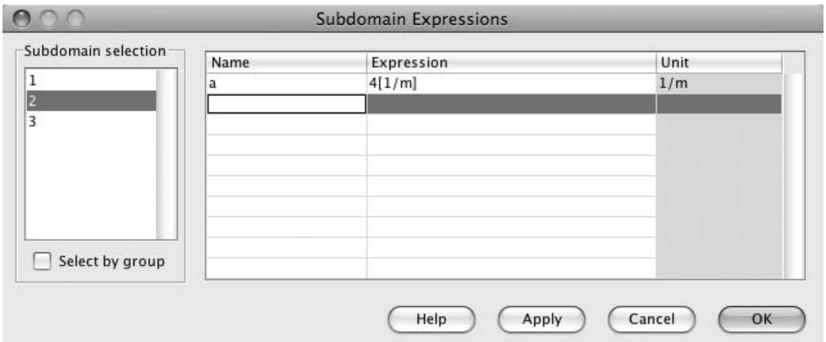


FIGURE 10.14 2D_Bio_TLI_1 model Subdomain Expressions (2) edit window

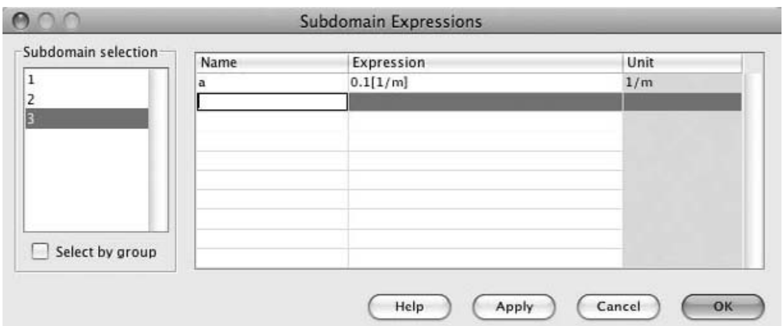


FIGURE 10.15 2D_Bio_TLI_1 model Subdomain Expressions (3) edit window

Table 10.3 Subdomain Settings

Name	Subdomain 1	Subdomain 2	Subdomain 3
k (isotropic)	k_tissue	k_tumor	k_skin
ρ	rho_tissue	rho_tumor	rho_skin
C	C_tissue	C_tumor	C_skin
ρ_b	rho_blood	rho_blood	rho_blood
C_b	C_blood	C_blood	C_blood
ω_b	wb_tissue	wb_tumor	wb_skin
T_b	T_blood	T_blood	T_blood
Q_{met}	Q_met	Q_met	Q_met
Q_{ext}	Q_laser	Q_laser	Q_laser

Physics Subdomain Settings: Bioheat Equation (htbh)

Having established the basic Physics settings for the 2D_Bio_TLI_1 model, the next step is to define the fundamental Physics subdomain setting. Using the menu bar, select Physics > Subdomain Settings. In the Subdomain Settings edit window, enter the information shown in Table 10.3 and Figures 10.16, 10.17, and 10.18. Click the Apply button after filling in the parameters of each separate subdomain in the subdomain edit window.

FIGURE 10.16 2D_Bio_TLI_1 model Subdomain Settings (1) edit window

FIGURE 10.17 2D_Bio_TLI_1 model Subdomain Settings (2) edit window

FIGURE 10.18 2D_Bio_TLI_1 model Subdomain Settings (3) edit window

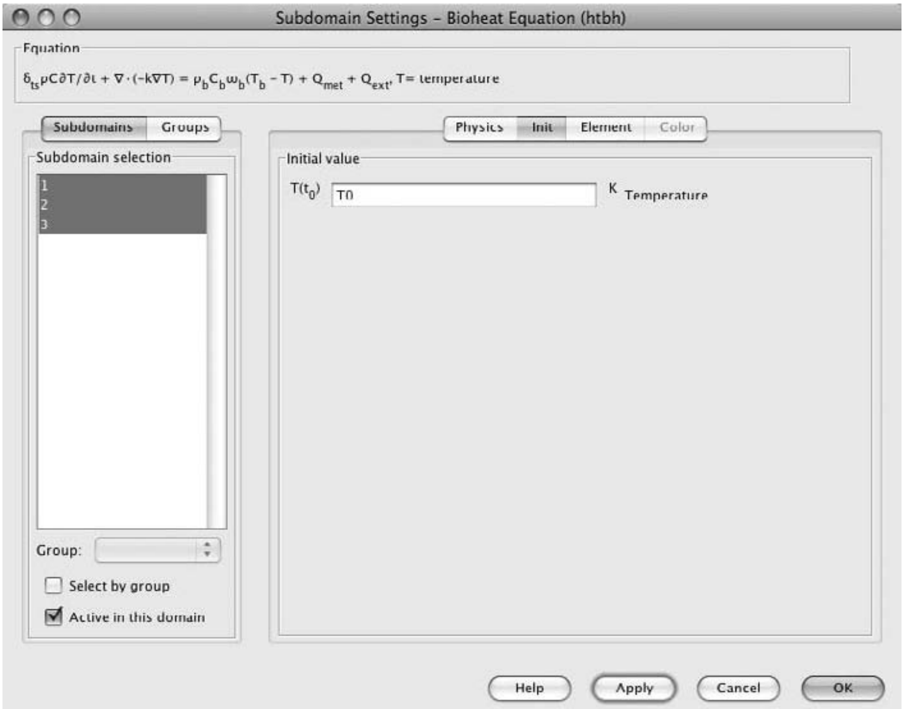


FIGURE 10.19 2D_Bio_TLl_1 model Subdomain Settings, Initial value edit window

Select subdomains 1, 2, and 3. Click the Init tab. Enter T0 in the Initial value edit window. Click the Apply button. See Figure 10.19. Click OK.

Physics Boundary Settings: Bioheat Equation (htbh)

Having established the subdomain settings for the 2D_Bio_TL1_1 model, the next step is to define the fundamental Physics boundary settings. Using the menu bar, Select Physics > Boundary Settings. In the Boundary Settings edit window, enter the information shown in Table 10.4. Click the Apply button after choosing or entering the

Table 10.4 Boundary Settings

Boundary	Boundary Condition	Parameter	Value	Figure Number
1, 3–5	Axial symmetry	—	—	10.20
2, 8, 9	Thermal insulation	—	—	10.21
7	Heat flux	h	h_conv	10.22
		T_{inf}	T_inf	

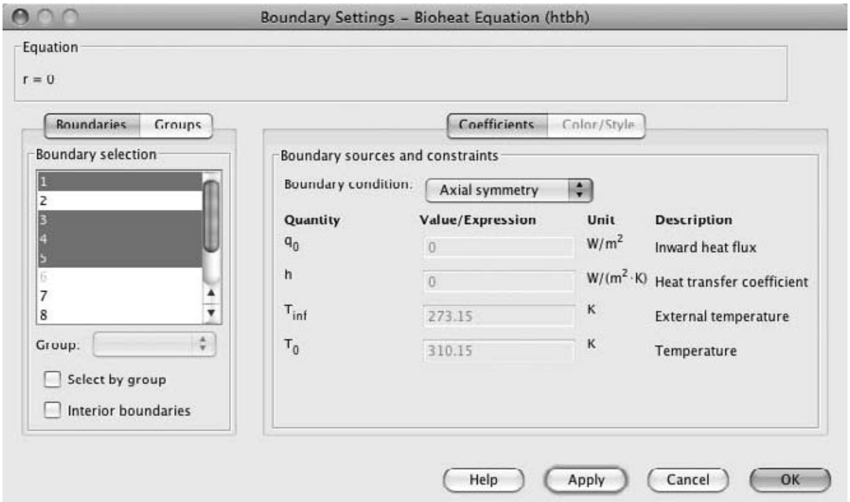


FIGURE 10.20 2D_Bio_TLI_1 model Boundary Settings (1, 3–5) edit window

parameters of each Boundary Settings group in the Boundary Settings edit window. Click OK. See Figures 10.20, 10.21, and 10.22.

Mesh Generation

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 2 (the tumor). Enter 0.005 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 10.23.

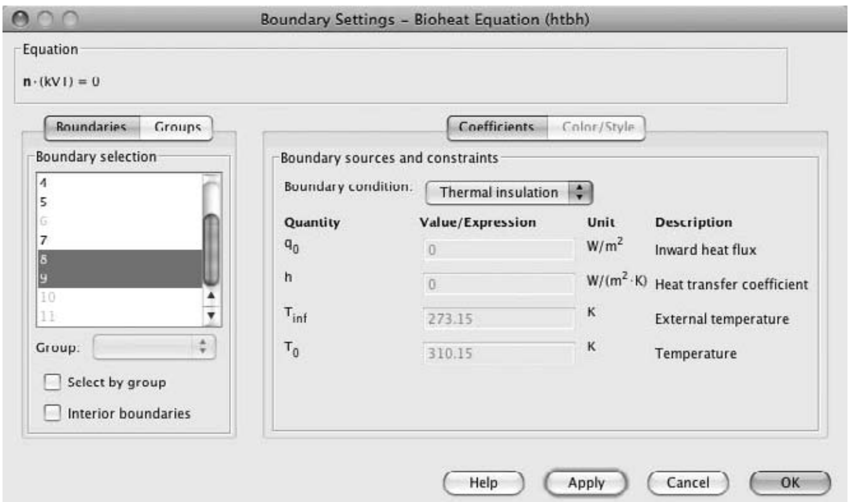


FIGURE 10.21 2D_Bio_TLI_1 model Boundary Settings (2, 8, 9) edit window

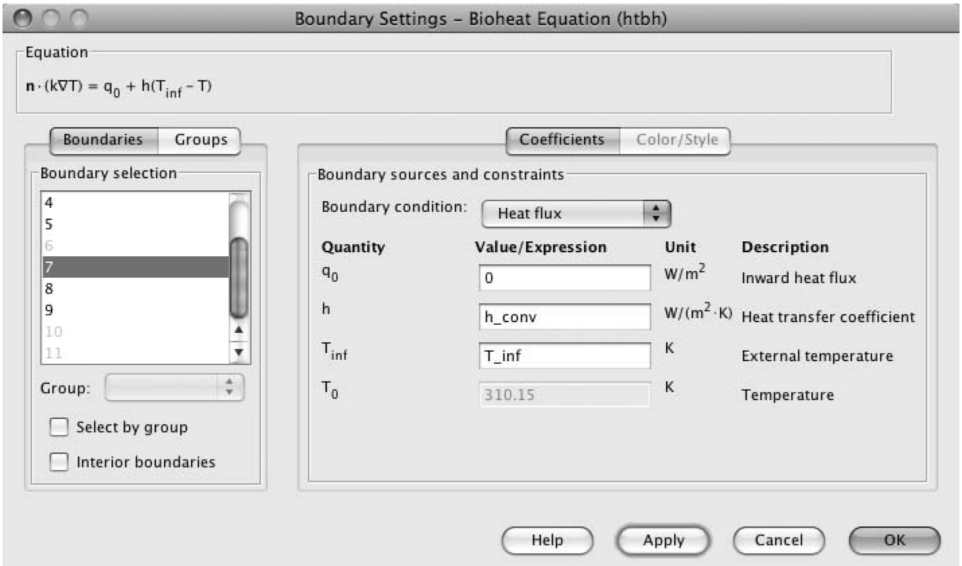


FIGURE 10.22 2D_Bio_TLI_1 model Boundary Settings (7) edit window

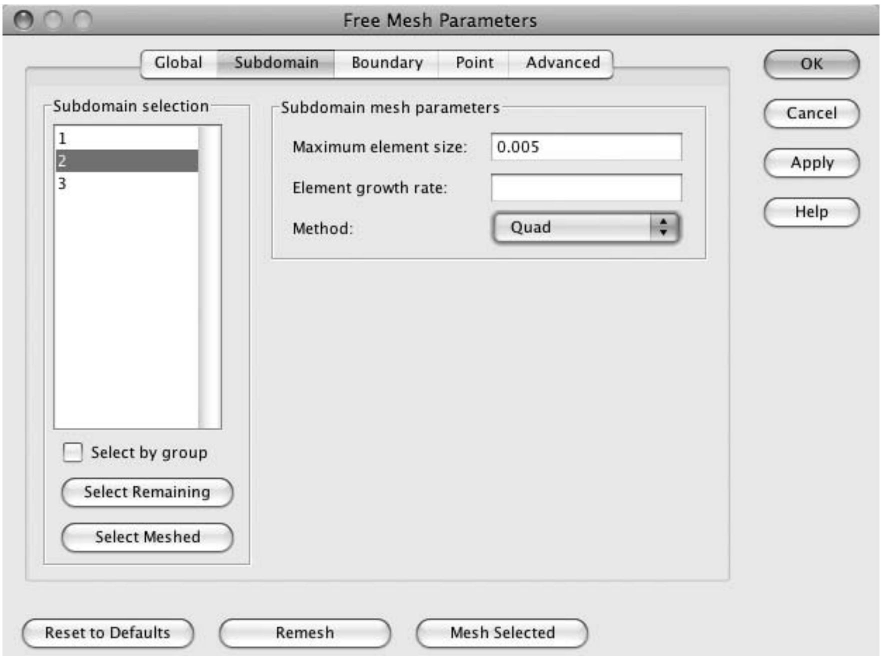


FIGURE 10.23 2D_Bio_TLI_1 model Free Mesh Parameters, subdomain 2 edit window

FIGURE 10.24 2D_Bio_TLI_1 model mesh

Click the Mesh Selected button. Click the Select Remaining button. Click the Mesh Selected button. Click OK. See Figure 10.24.

Solving the 2D_Bio_TLI_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Time dependent" in the Solver list. Enter 0:10:600 in the Times edit window. See Figure 10.25.

Click OK. Using the menu bar, select Solve > Solve Problem.

Postprocessing and Visualization

The default plot shows a surface plot of the temperature (K). See Figure 10.26.

Typically, such analytical plots are viewed in degrees Centigrade. The plot can be easily converted through the following steps. Using the menu bar, Select

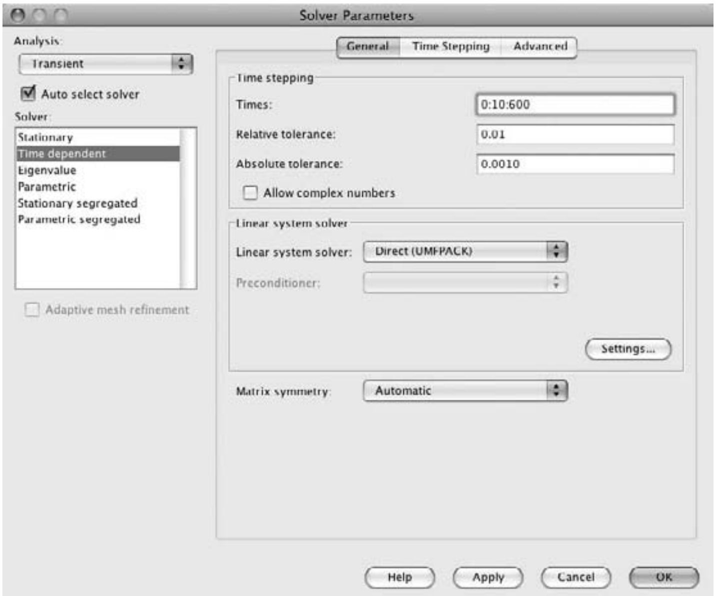


FIGURE 10.25 2D_Bio_TLI_1 model Solver Parameters edit window

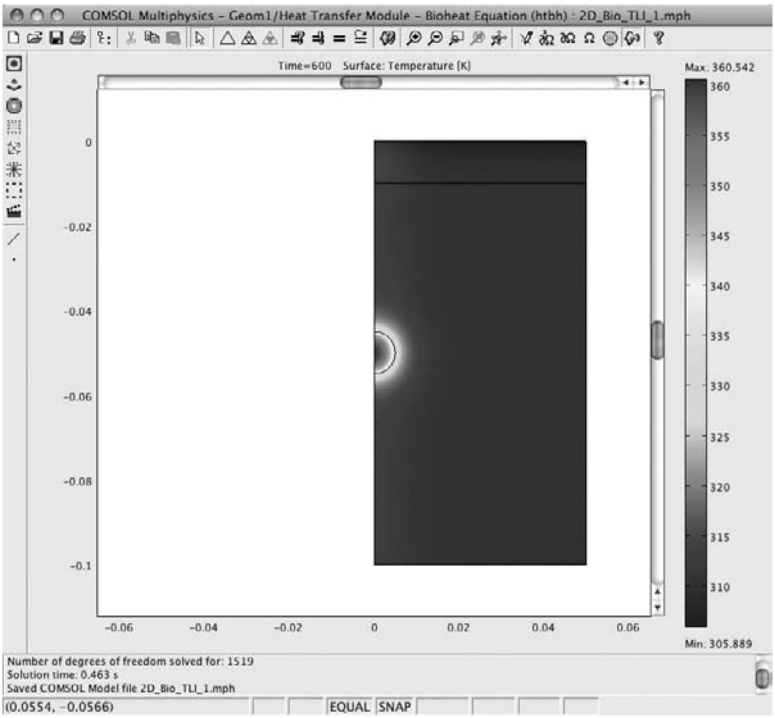


FIGURE 10.26 2D_Bio_TLI_1 model solution, temperature (K)

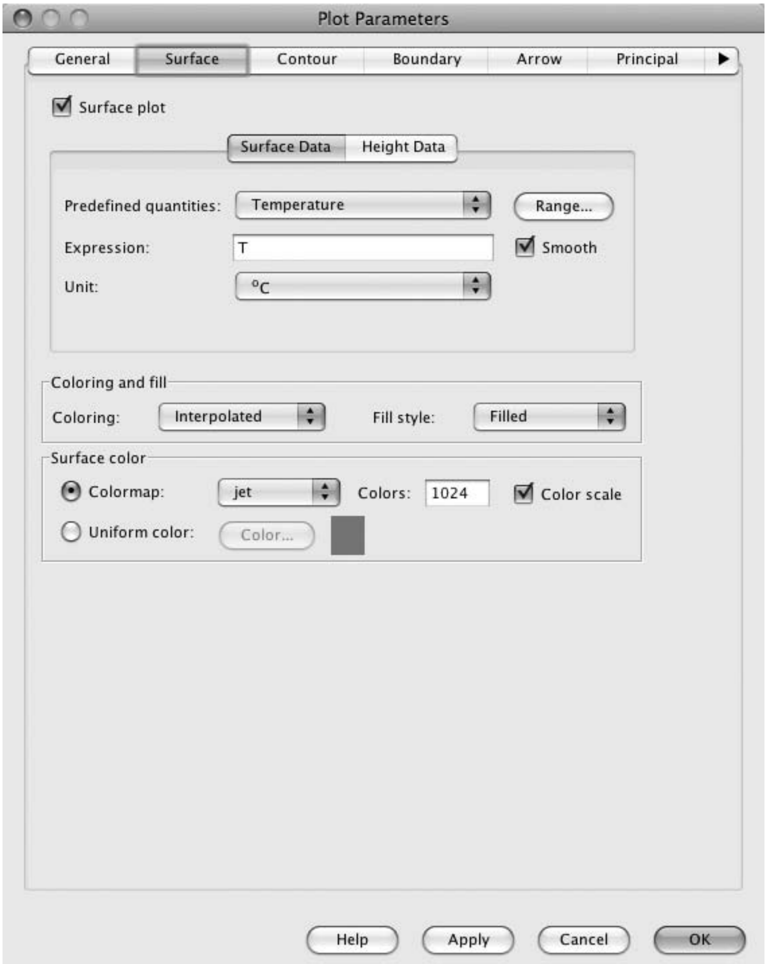


FIGURE 10.27 2D_Bio_TLI_1 model Plot Parameters, Surface tab (°C) edit window

Postprocessing > Plot Parameters > Surface. Select "degC (°C)" from the Unit pull-down list. See Figure 10.27.

Click OK. See Figure 10.28.

NOTE The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. The guiding principle needs to be that tumor cells die at elevated temperatures. The literature cites temperatures that range from 42 °C (315.15 K)[5] to 60 °C (333.15 K).[6] If the postulated method raises the local temperature of the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful.

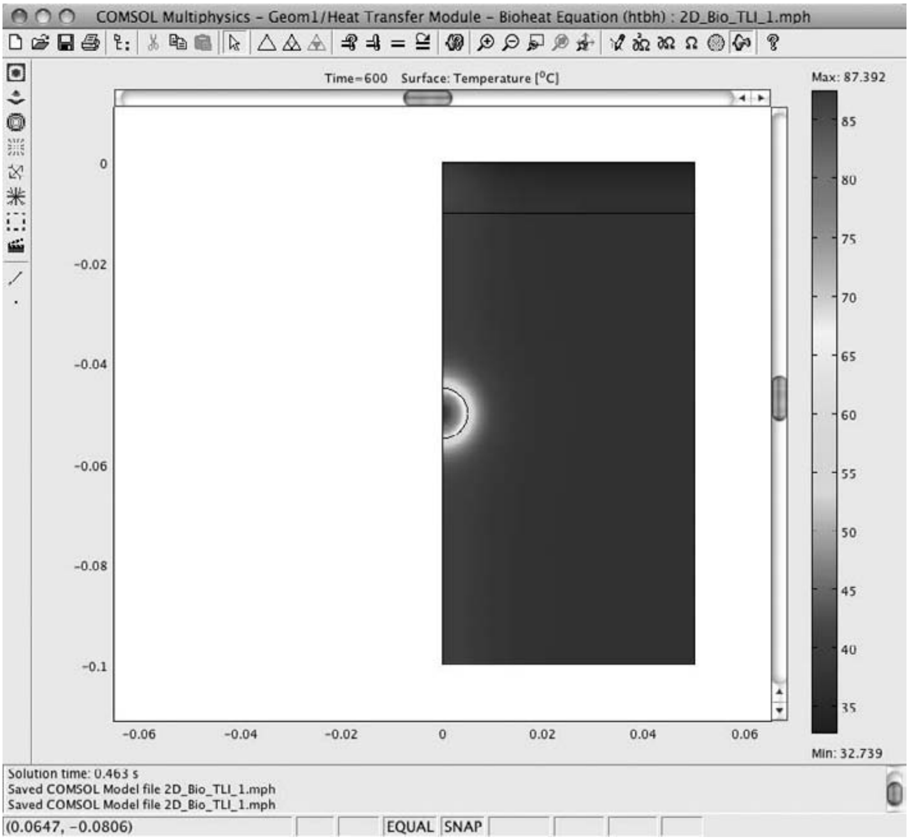


FIGURE 10.28 2D_Bio_TLI_1 model, surface temperature (°C)

Now that the 2D_Bio_TLI_1 model has been successfully calculated, the modeler can determine the time to reach the desired temperature for a preliminary estimate of an effective treatment. Final determination of the veracity of the calculation will, of course, need to be made experimentally. The results (estimated time values) from the model calculations will significantly reduce the effort needed to determine an accurate experimental value.

To determine the time to the desired temperature of 60 °C at the boundary of the tumor, proceed as follows. Using the menu bar, select Postprocessing > Domain Plot Parameters. Click the Point tab. Select point 6 in the Point selection window. Select "Temperature" from the Predefined quantities pull-down list. Select "degC (°C)" from the Unit pull-down list. See Figure 10.29.

Click OK. Figure 10.30 shows that the time to 60 (C at the boundary of the tumor is approximately 220 seconds under the specified conditions of this model).

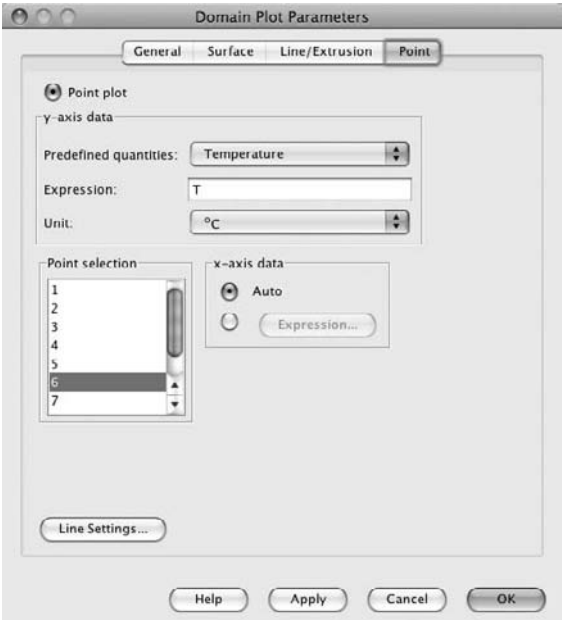


FIGURE 10.29 2D_Bio_TLI_1 model Domain Plot Parameters, Point tab

FIGURE 10.30 2D_Bio_TLI_1 model, time to temperature

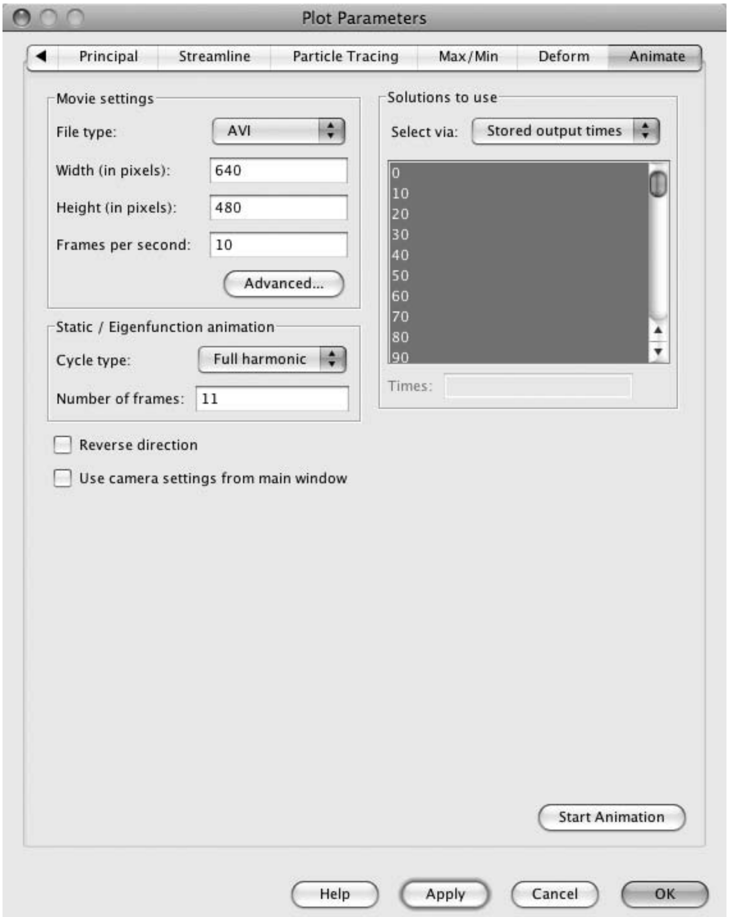


FIGURE 10.31 2D_Bio_TLI_1 model Plot Parameters, Animate tab

The development of the entire model can be viewed by following these steps. Using the menu bar, select Postprocessing > Plot Parameters > Animate. Select all solutions in the Solutions to use window. See Figure 10.31.

Click the Start Animation button. See Figure 10.32.

2D Axisymmetric Tumor Laser Irradiation Model: Summary and Conclusions

The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. Now that the 2D_Bio_TLI_1 model has been successfully calculated, the modeler can determine the time to reach the desired temperature for a preliminary estimate of an effective treatment. Final determination of the veracity of

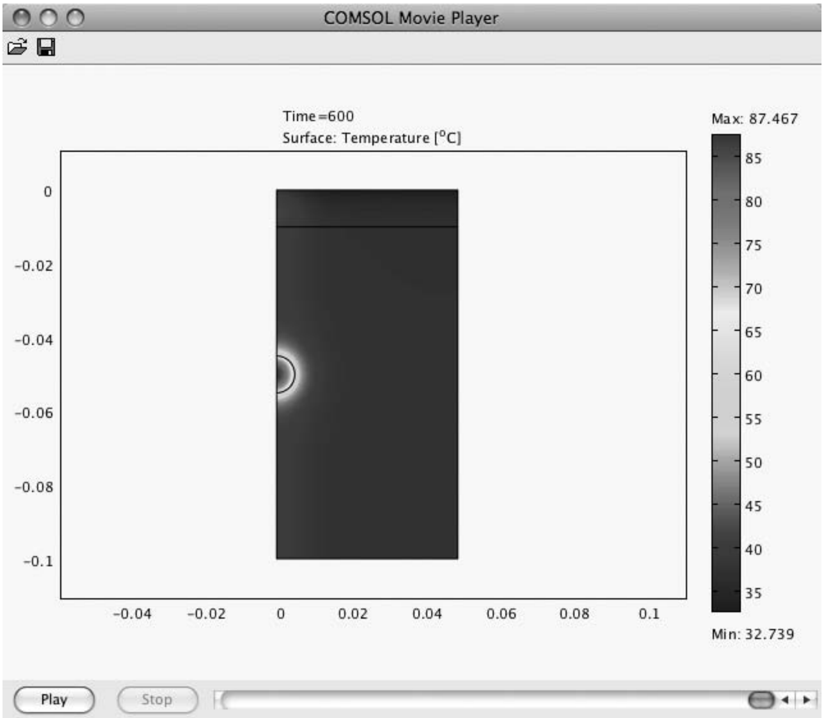


FIGURE 10.32 2D_Bio_TLI_1 model Plot Parameters, animation, final frame

the calculation will, of course, need to be made experimentally. The literature cites temperatures that range from 42 °C (315.15 K) to 60 °C (333.15 K).[6]

If the postulated method raises the local temperature of the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful. The results (estimated time values) from the model calculations will significantly reduce the effort needed to determine an accurate experimental value. The guiding principle needs to be that tumor cells die at elevated temperatures.

■ Microwave Cancer Therapy Theory

Hyperthermic (high-temperature) oncology (cancer, tumor)[8] involves the use of elevated temperatures to kill cancer and other tumor cells. As discussed in the previous model, it is necessary to locally raise the temperature of the cancer/tumor cells, without doing significant damage to the normal (healthy) cells surrounding the tumor. In the previous model, the energy was supplied as photothermal energy using laser irradiation. In this model, the externally applied energy is supplied through the use of a

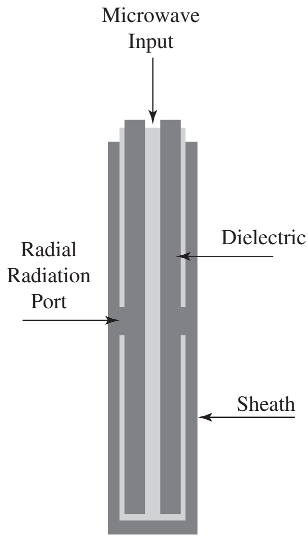


FIGURE 10.33 2D_Bio_MCT_1 model, antenna

specialized microwave antenna and the application of Ohm's and Joule's Laws.[9,10] This type of procedure is typically designated as a minimally invasive procedure.[11]

Figure 10.33 shows the microwave antenna in cross section.

2D Axisymmetric Microwave Cancer Therapy Model

The following numerical solution model (2D_Bio_MCT_1 model) is derived from a model that was originally developed by COMSOL as a Heat Transfer Module tutorial model for the demonstration of the solution of a bioheat equation model. That model was developed for distribution with the Heat Transfer Module software as part of the COMSOL Heat Transfer Module Model Library.

NOTE The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. The guiding principle needs to be that tumor cells die at elevated temperatures. The literature cites temperatures that range from 42 °C (315.15 K)[5] to 60 °C (333.15 K).[6] If the postulated method raises the local temperature of the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful.

This model takes advantage of the conductivity of human tissue. Figure 10.34 shows the microwave antenna in cross section, embedded in the modeling domain (tissue) and radiating power. Because the model is created as a 2D axisymmetric model, only the right half of the structure is used in the calculations.

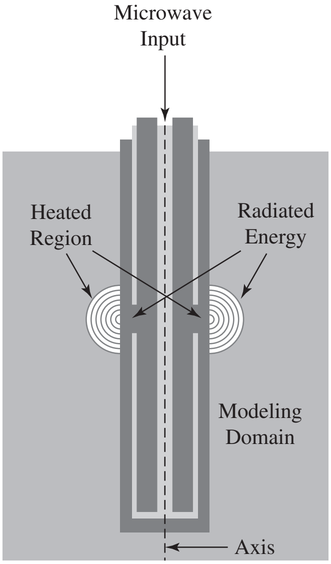


FIGURE 10.34 2D_Bio_MCT_1 model, antenna plus tissue

To start building the 2D_Bio_MCT_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > Bioheat Equation > Steady-state analysis. Click the Multiphysics button, and then click the Add button. See Figure 10.35.

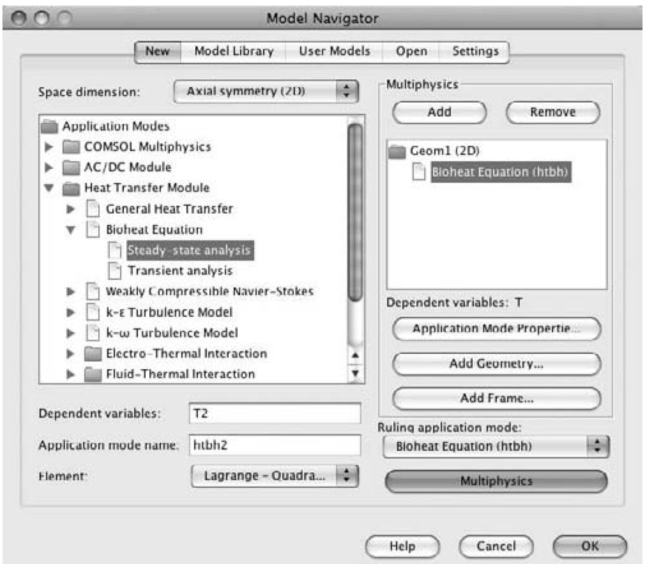


FIGURE 10.35 2D_Bio_MCT_1 Model Navigator, Heat Transfer Module

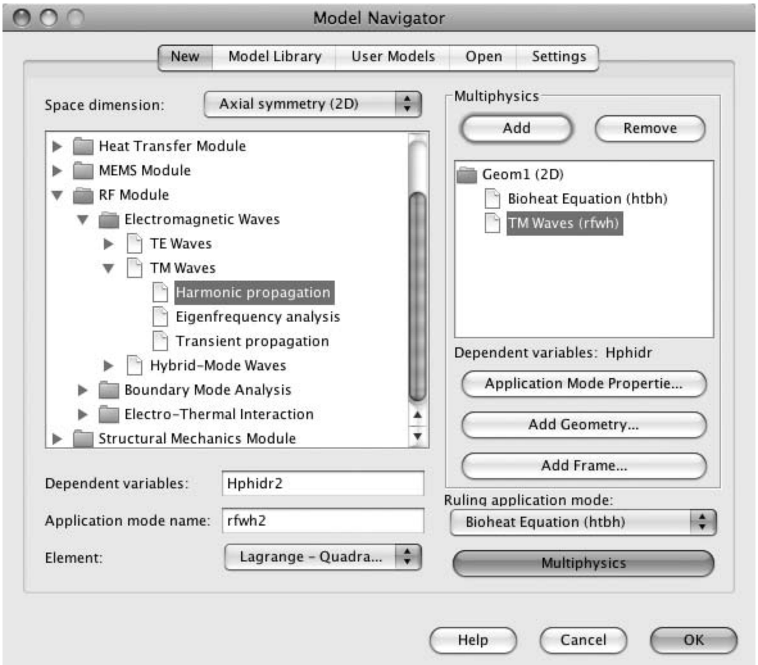


FIGURE 10.36 2D_Bio_MCT_1 Model Navigator, RF Module

Select RF Module > Electromagnetic Waves > TM Waves > Harmonic propagation. Select "Lagrange-Quartic" from the Element pull-down list. Click the Add button. See Figure 10.36. Click OK.

NOTE To verify the Lagrange-Quartic choice, the modeler can at any time go to the menu bar, select "Model Navigator," select the Application mode of choice, click on the Application Mode Properties button, and then verify the setting of choice.

Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 10.5; see also Figure 10.37. Click OK.

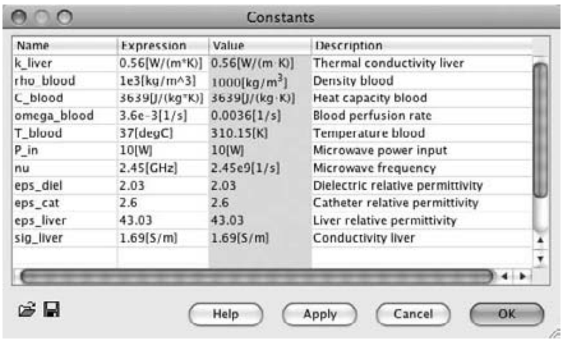
Using the menu bar, select File > Save As. Enter 2D_Bio_MCT_1.mph in the Save As edit window. See Figure 10.38. Click the Save button.

Geometry Modeling

Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 10.6. Click OK after filling in the parameters of each separate rectangle in the Rectangle edit window. See Figures 10.39 and 10.40.

Table 10.5 Constants Edit Window

Name	Expression	Description
k_liver	0.56[W/(m*K)]	Thermal conductivity liver
rho_blood	1e3[kg/m^3]	Density blood
C_blood	3639[J/(kg*K)]	Heat capacity blood
omega_blood	3.6e-3[1/s]	Blood perfusion rate
T_blood	37[degC]	Temperature blood
P_in	10[W]	Microwave power input
nu	2.45[GHz]	Microwave frequency
eps_diel	2.03	Dielectric relative permittivity
eps_cat	2.6	Catheter relative permittivity
eps_liver	43.03	Liver relative permittivity
sig_liver	1.69[S/m]	Conductivity liver



Name	Expression	Value	Description
k_liver	0.56[W/(m*K)]	0.56[W/(m·K)]	Thermal conductivity liver
rho_blood	1e3[kg/m^3]	1000[kg/m^3]	Density blood
C_blood	3639[J/(kg*K)]	3639[J/(kg·K)]	Heat capacity blood
omega_blood	3.6e-3[1/s]	0.0036[1/s]	Blood perfusion rate
T_blood	37[degC]	310.15[K]	Temperature blood
P_in	10[W]	10[W]	Microwave power input
nu	2.45[GHz]	2.45e9[1/s]	Microwave frequency
eps_diel	2.03	2.03	Dielectric relative permittivity
eps_cat	2.6	2.6	Catheter relative permittivity
eps_liver	43.03	43.03	Liver relative permittivity
sig_liver	1.69[S/m]	1.69[S/m]	Conductivity liver

FIGURE 10.37 2D_Bio_MCT_1 model Constants edit window

FIGURE 10.38 2D_Bio_MCT_1 model Save As edit window

Table 10.6 Geometry Components

Name	Width	Height	Base	r	z	Figure Number
R1	0.595e-3	0.01	Corner	0	0	10.39
R2	29.405e-3	0.08	Corner	0.595e-3	1.25e-30	10.40

Click the Zoom Extents button. See Figure 10.41.

Using the menu bar, select Draw > Create Composite Object. Enter R1+R2 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 10.42.

Click OK. See Figure 10.43.

NOTE The composite object (CO1) created through these steps forms the modeling domain that constitutes the liver tissue.

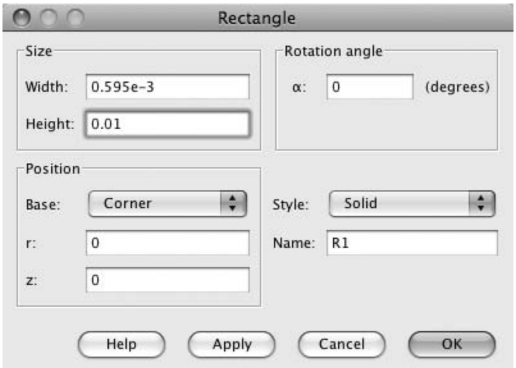


FIGURE 10.39 2D_Bio_MCT_1 model Rectangle (R1) edit window

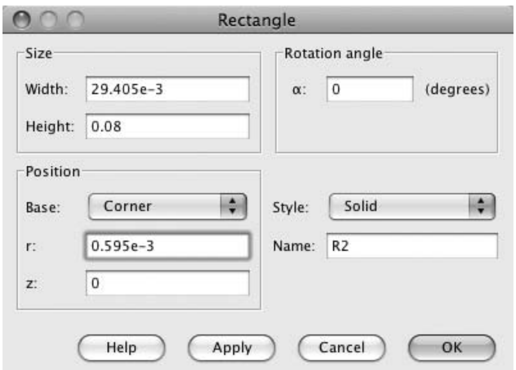


FIGURE 10.40 2D_Bio_MCT_1 model Rectangle (R2) edit window

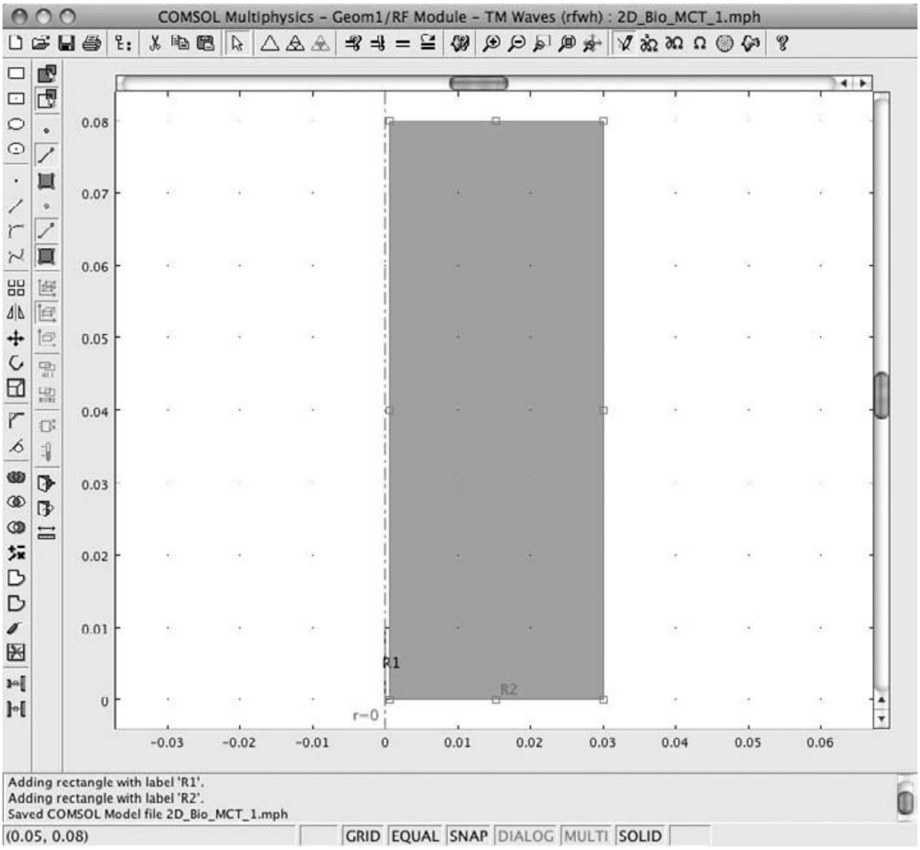


FIGURE 10.41 2D_Bio_MCT_1 model rectangles (R1, R2)

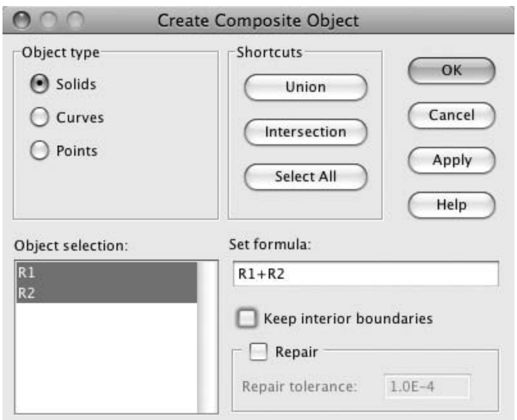


FIGURE 10.42 2D_Bio_MCT_1 model Create Composite Object edit window

FIGURE 10.43 2D_Bio_MCT_1 model composite object (CO1)

Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 10.7. Click OK after filling in the parameters of each separate rectangle in the Rectangle edit window. See Figures 10.44 and 10.45.

Using the menu bar, select Draw > Create Composite Object. Enter R1+R2 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 10.46. Click OK.

Table 10.7 Geometry Components

Name	Width	Height	Base	r	z	Figure Number
R1	0.125e-3	1.0e-3	Corner	0.47e-3	0.0155	10.44
R2	3.35e-4	0.0699	Corner	0.135e-3	0.0101	10.45

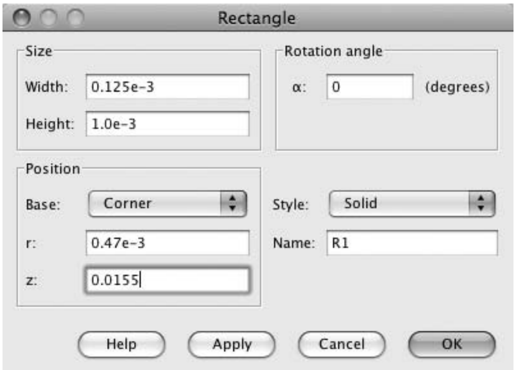


FIGURE 10.44 2D_Bio_MCT_1 model Rectangle (R1) edit window

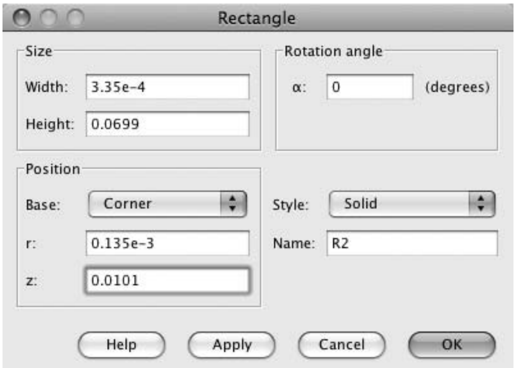


FIGURE 10.45 2D_Bio_MCT_1 model Rectangle (R2) edit window

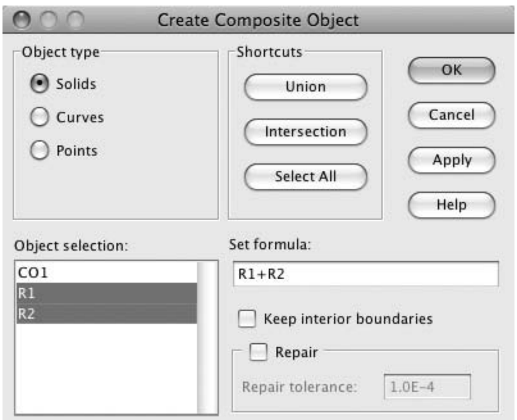


FIGURE 10.46 2D_Bio_MCT_1 model Create Composite Object edit window

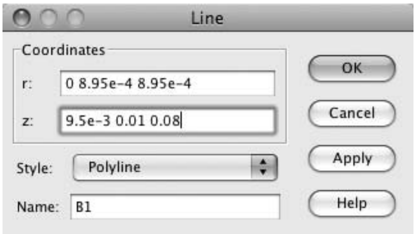


FIGURE 10.47 2D_Bio_MCT_1 model Line (B1) edit window

Table 10.8 Geometry Components

Name	Width	Height	Base	r	z	Figure Number
R3	1.25e-4	1.0e-3	Corner	4.7e-4	0.0155	10.48

NOTE The composite object created through these steps (CO2) forms the modeling domain that constitutes the antenna dielectric.

Next, add a line to the geometry. Using the menu bar, select Draw > Specify Objects > Line. In the r edit window, enter 0 8.95e-4 8.95e-4. In the z edit window, enter 9.5e-3 0.01 0.08. See Figure 10.47. Click OK.

NOTE The line created through these steps forms the boundary of the antenna sheath.

Add the last rectangle to the geometry. Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 10.8. Click OK after filling in the parameters of the rectangle in the Rectangle edit window. See Figure 10.48.

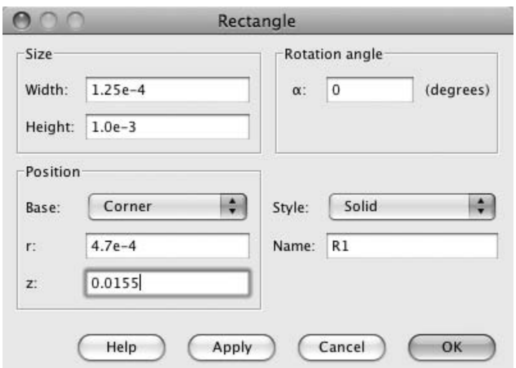


FIGURE 10.48 2D_Bio_MCT_1 model Rectangle (R1) edit window

FIGURE 10.49 2D_Bio_MCT_1 model, antenna and model domain

NOTE The rectangle created through these steps forms the modeling domain that constitutes the slot in the coaxial antenna that allows energy to be radiated into the liver tissue.

Click OK. See Figure 10.49.

Having established the geometry for the 2D_Bio_MCT_1 model, the next step is to define the fundamental Physics properties.

Physics Settings: Bioheat Equation (htbh)

Using the menu bar, select Multiphysics > 1 Bioheat Equation (htbh). Select Physics > Subdomain Settings. Select subdomains 2, 3, and 4. Uncheck the Active in this domain check box. See Figure 10.50. Click the Apply button.

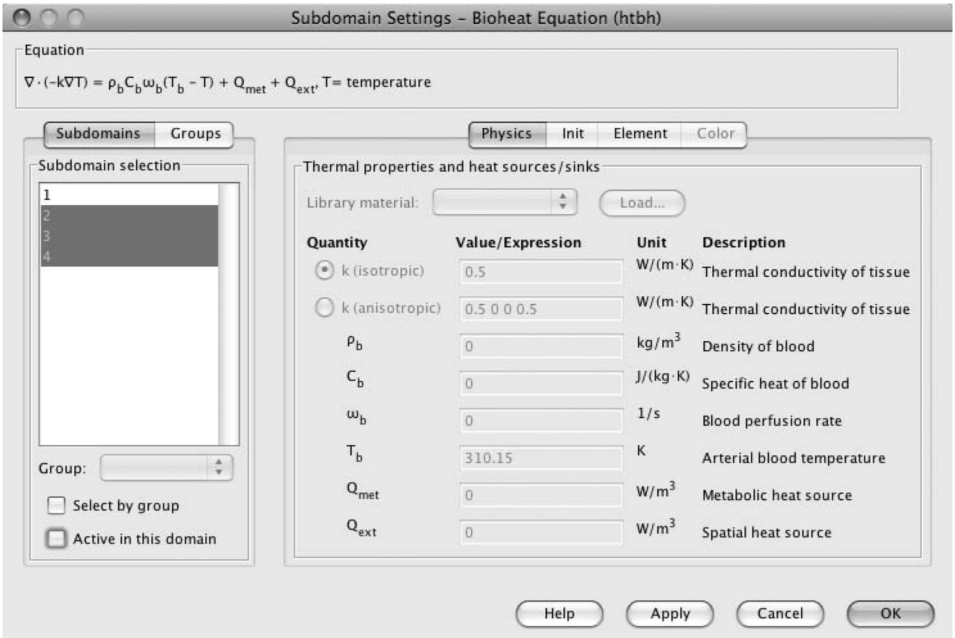


FIGURE 10.50 2D_Bio_MCT_1 model Subdomain Settings (2, 3, 4) edit window

Select subdomain 1. In the subdomain 1 edit window, enter the information as shown in Table 10.9. Click OK after filling in the parameters of the rectangle in the Rectangle edit window.

NOTE The metabolic energy (Q_{met}) is sufficiently small, relative to the microwave energy, that it can be ignored in this model. Thus it is set to zero.

Click the Apply button. See Figure 10.51. Click OK.

Table 10.9 Subdomain 1 Settings

Name	Setting
k (isotropic)	k_liver
ρ_b	rho_blood
C_b	C_blood
ω_b	omega_blood
T_b	T_blood
Q_{met}	0
Q_{ext}	Qav_rfwh

FIGURE 10.51 2D_Bio_MCT_1 model Subdomain Settings (1) edit window

Physics Boundary Settings: Bioheat Equation (htbh)

Having established the bioheat equation subdomain settings for the 2D_Bio_MCT_1 model, the next step is to define the bioheat equation physics boundary settings.

Using the menu bar, select Physics > Boundary Settings. Select boundary 1. Check the Select by group check box.

Select "Thermal insulation" from the Boundary condition pull-down list. See Figure 10.52. Click OK.

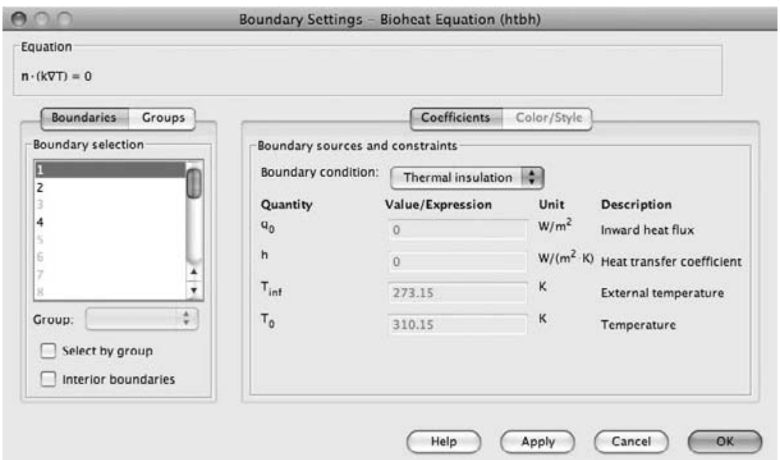


FIGURE 10.52 2D_Bio_MCT_1 model Boundary Settings (1) edit window

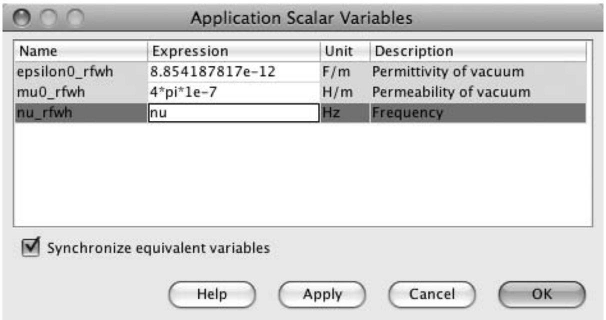


FIGURE 10.53 2D_Bio_MCT_1 model Application Scalar Variables

NOTE The thermal insulation boundary condition can be employed because most of the heat energy is removed by perfusion, rather than by conduction through the boundaries.

Physics Settings: 2 TM Waves (rfwh), Scalar Variables

Using the menu bar, select Multiphysics > 2 TM Waves (rfwh). Using the menu bar, select Physics > Scalar Variables. Enter nu in the nu_rfwh (Frequency) edit window. See Figure 10.53. Click OK.

Physics Settings: 2 TM Waves (rfwh), Subdomain Settings

Using the menu bar, select Physics > Subdomain Settings. In the Subdomain Settings edit window, enter the information shown in Table 10.10 and Figures 10.54 through 10.57. Click the Apply button after filling in the parameters of each separate subdomain in the Subdomain settings edit window. Click OK.

Physics Settings: 2 TM Waves (rfwh), Boundary Settings

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings edit window, enter the information as shown in Table 10.11. Click the Apply button after choosing or entering the parameters of each Boundary Settings group in the Boundary Settings edit windows. Click OK. See Figures 10.58–10.62.

Table 10.10 Subdomain Settings

Name	Subdomain 1	Subdomain 2	Subdomain 3	Subdomain 4
ε_r (isotropic)	eps_liver	eps_cat	eps_diel	1
σ (isotropic)	sig_liver	0	0	0
μ_r	1	1	1	1

Table 10.11 Boundary Settings

Boundary	Boundary Condition	Wave Type	Value	Figure Number
1, 3	Axial symmetry	—	—	10.58
2, 14, 18, 20, 21	Scattering boundary condition	Spherical	—	10.59
5–7, 9, 11–13, 15, 17	Perfect electric conductor	—	—	10.60
8	Port	Wave excitation selected	P_in	10.61
8	Port tab	—	Coaxial	10.62

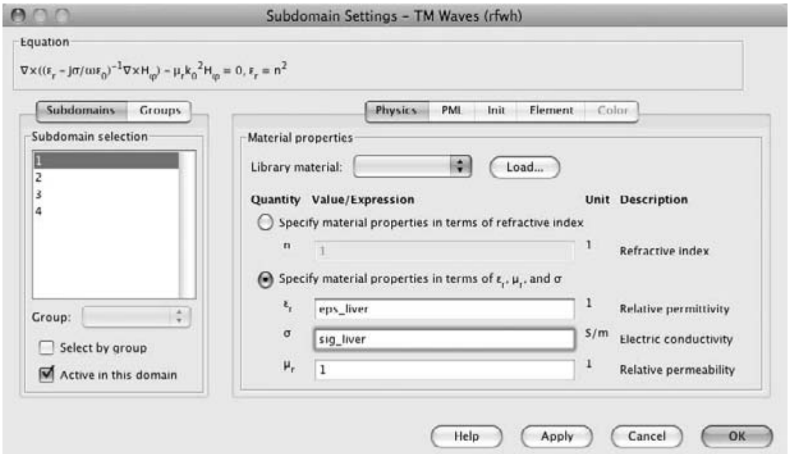


FIGURE 10.54 2D_Bio_MCT_1 model, Subdomain Settings (1) edit window

FIGURE 10.55 2D_Bio_MCT_1 model, Subdomain Settings (2) edit window

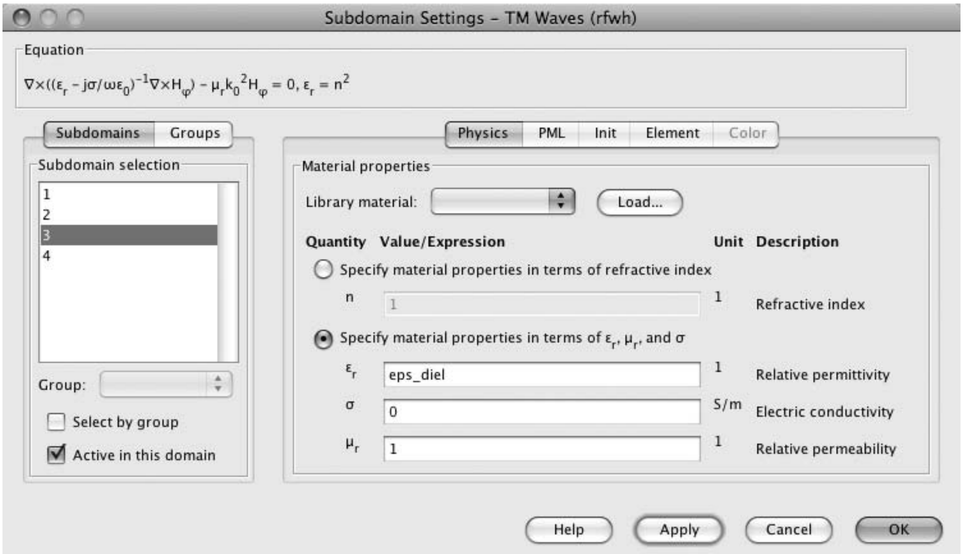


FIGURE 10.56 2D_Bio_MCT_1 model, Subdomain Settings (3) edit window

Mesh Generation

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Custom mesh size radio button. Enter 3e-3 in the Maximum element size edit window. Click the Apply button. See Figure 10.63.

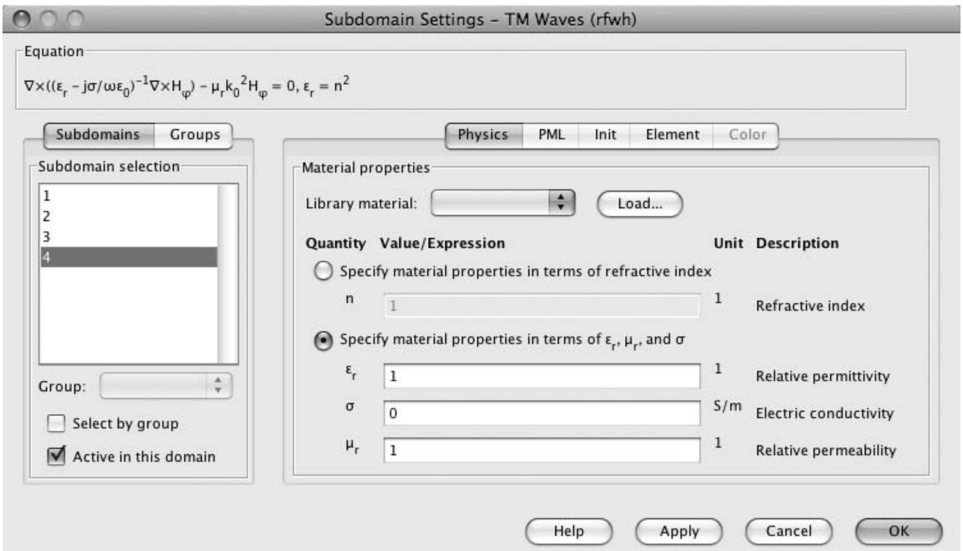


FIGURE 10.57 2D_Bio_MCT_1 model, Subdomain Settings (4) edit window

FIGURE 10.58 2D_Bio_MCT_1 model Boundary Settings (1, 3) edit window

FIGURE 10.59 2D_Bio_MCT_1 model Boundary Settings (2, 14, 18, 20, 21) edit window

FIGURE 10.60 2D_Bio_MCT_1 model Boundary Settings (5–7, 9, 11–13, 15, 17) edit window

FIGURE 10.61 2D_Bio_MCT_1 model Boundary Settings (8) edit window

FIGURE 10.62 2D_Bio_MCT_1 model Boundary Settings (8), Port tab edit window

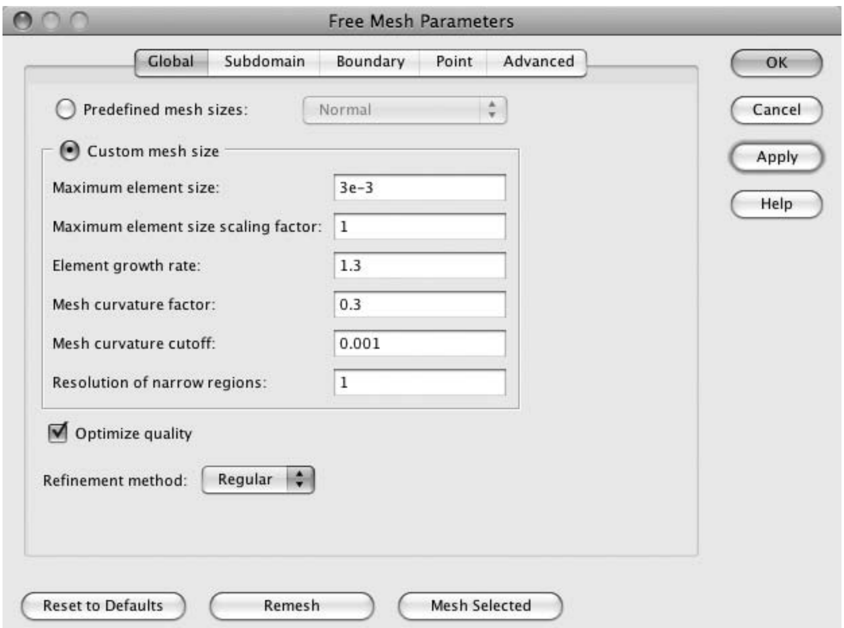


FIGURE 10.63 2D_Bio_MCT_1 model Free Mesh Parameters, Global tab

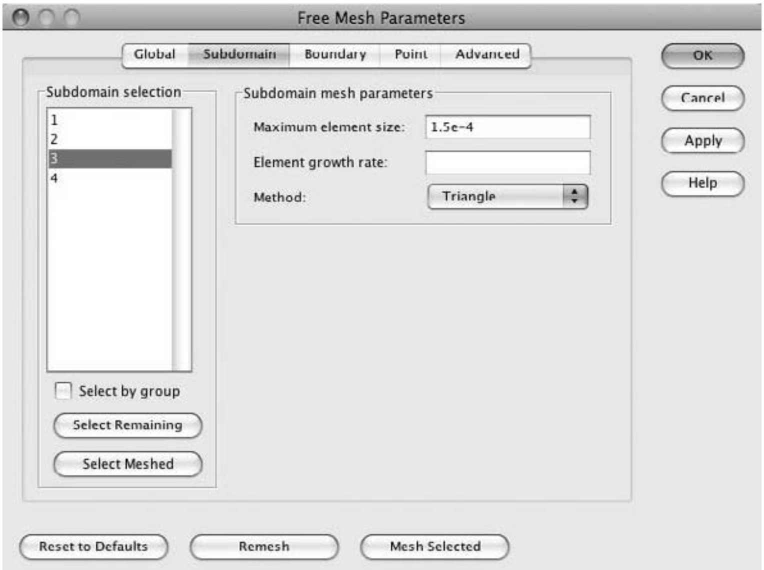


FIGURE 10.64 2D_Bio_MCT_1 model Free Mesh Parameters, subdomain 3

Click the Subdomain tab. Select subdomain 3. Enter 1.5e-4 in the Maximum element size edit window. See Figure 10.64.

Click the Remesh button, and then click OK. See Figure 10.65.

Solving the 2D_Bio_MCT_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Parametric" from the solver list. Enter P_in in the Parameter name edit window. Enter 2:0.5:10 in the Parameter values edit window. Click the Apply button. See Figure 10.66.

Click OK. Using the menu bar, select Solve > Solve Problem.

Postprocessing and Visualization

The default plot shows a surface plot of the temperature (K). See Figure 10.67.

Typically, such analytical plots are viewed in degrees Centigrade. The plot can be easily converted through the following steps. Using the menu bar, select Postprocessing > Plot Parameters > Surface. Select "degC (°C)" from the Unit pull-down list. See Figure 10.68.

Click OK. See Figure 10.69.

NOTE The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. The guiding principle needs to be that tumor cells die at elevated temperatures. The literature cites temperatures that range from 42 °C (315.15 K) to 60 °C (333.15 K).[6] If the postulated method raises the local temperature of

FIGURE 10.65 2D_Bio_MCT_1 model mesh

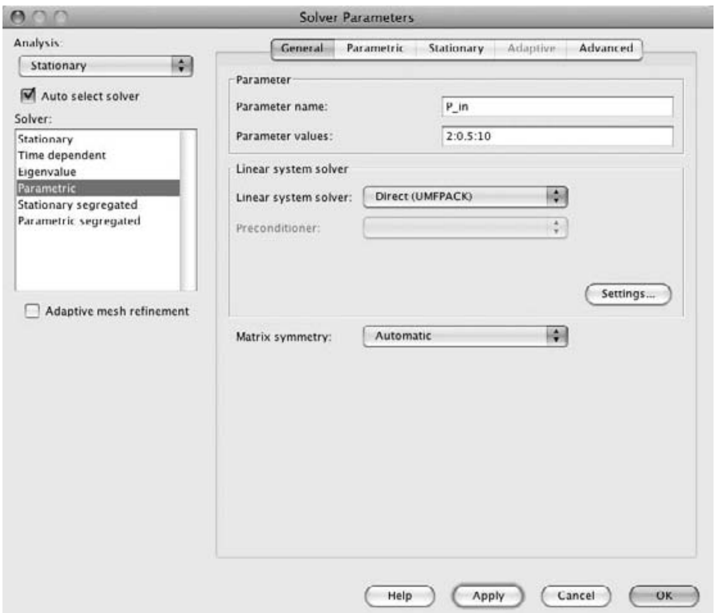


FIGURE 10.66 2D_Bio_MCT_1 model Solver Parameters

FIGURE 10.67 2D_Bio_MCT_1 model solution, temperature (K)

the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful.

Now that the 2D_Bio_MCT_1 model has been successfully calculated, the modeler can determine the temperature for a preliminary estimate of the input power. Final determination of the veracity of the calculation will, of course, need to be made experimentally. The results (solutions at different input powers) from the model calculations will significantly reduce the effort needed to determine an accurate initial experimental value.

It can readily be seen in the solution of the 2D_Bio_MCT_1 model at 10 W that the peak temperature in the region immediately adjacent to the antenna may be higher (approximately 100 °C) than desired. The range of solutions for powers from 2 W to 10 W is easily viewed for selection. Using the menu bar, select Postprocessing >

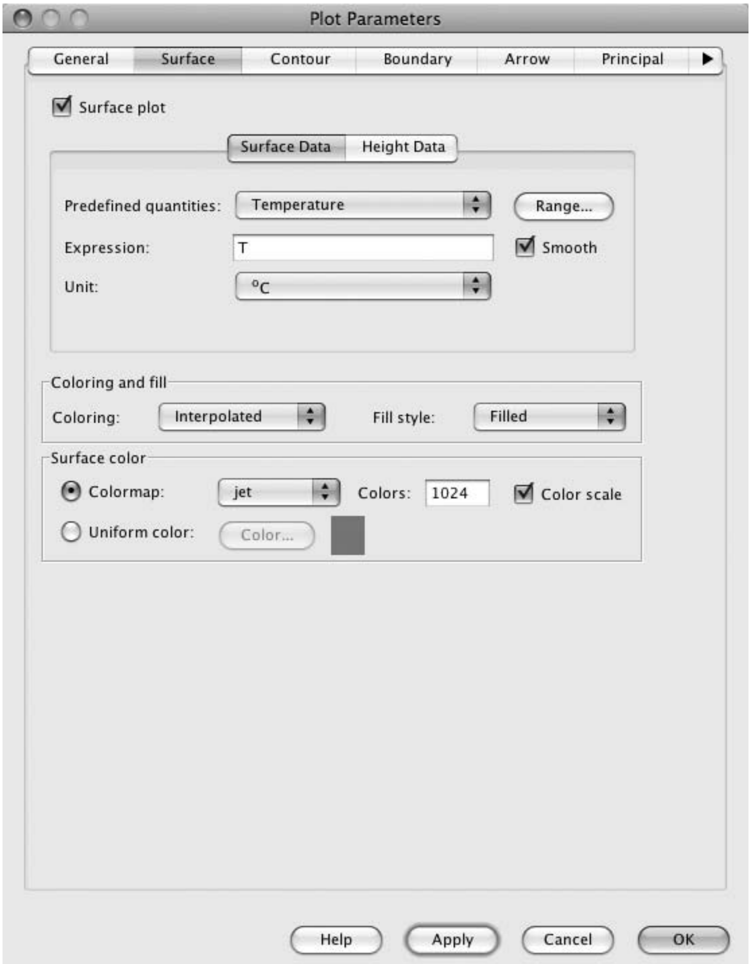


FIGURE 10.68 2D_Bio_MCT_1 model Plot Parameters, Surface tab (°C)

Cross-Section Plot Parameters > General. Select all of the solutions in the Solutions to use selection window. See Figure 10.70.

Click the Line/Extrusion tab. Select "degC (°C)" from the Unit pull-down list. Select "r" from the x-axis data pull-down list. Enter r0 = 0, and r1 = 0.03. Enter z0 = 0.02, and z1 = 0.02. See Figure 10.71.

Click the Apply button, and then click OK. See Figure 10.72.

Using the cross-section graph, an appropriate power/temperature/distance profile can be chosen for the desired therapy.

NOTE The plot lines of temperature on the cross-section graph are arranged in order of ascending power vertically. As more power is supplied to the tissue, more heat is

FIGURE 10.69 2D_Bio_MCT_1 model, surface temperature (°C)

FIGURE 10.70 2D_Bio_MCT_1 model Cross-Section Plot Parameters, General tab

FIGURE 10.71 2D_Bio_MCT_1 model Cross-Section Plot Parameters, Line/Extrusion tab

FIGURE 10.72 2D_Bio_MCT_1 model, temperature (T) as a function of applied power vs. radius (r)

dissipated in the tissue; accordingly, the temperature rises. The graphical plots start at 2 W and ascend to 10 W in 0.5-W increments.

2D Axisymmetric Microwave Cancer Therapy Model: Summary and Conclusions

The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. Now that the 2D_Bio_MCT_1 model has been successfully calculated, the modeler can determine the power needed to reach the desired temperature for a preliminary estimate of an effective treatment. Final determination of the veracity of the calculation will, of course, need to be made experimentally. The literature cites temperatures that range from 42 °C (315.15 K)[5] to 60 °C (333.15 K).[6]

If the postulated method raises the local temperature of the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful. The results (estimated power values) from the model calculations will significantly reduce the effort needed to determine an accurate experimental value. The guiding principle needs to be that tumor cells die at elevated temperatures.

■ References

1. H. H. Pennes, *J. Appl. Physiology*, Vol. 1, No. 2, August 1948, pp. 93–122.
2. http://en.wikipedia.org/wiki/Bioheat_transfer
3. http://medical-dictionary.thefreedictionary.com/perfusion
4. L. R. Hirsch et al., “Targeted Photothermal Tumor Therapy using Metal Nanoshells”, Proceedings of the Second Joint EMBS/BMES Conference, Houston, TX, USA, October 23–26, 2002, pp. 530–531.
5. L. R. Hirsch et al., *Proceedings of the 25th Annual International Conference of the IEEE EMBS,* Cancun, Mexico. September 17–21, 2003.
6. K. Saito et al., “Localized Heating by the Coaxial-Dipole Antenna for Microwave Coagulation Therapy”, Antennas, Propagation, and EM Theory, 2000. *Proceedings of ISAPE,* 2000, Beijing, China, 5th International Symposium.
7. D. P. O’Neal et al., “Photo-thermal tumor ablation in mice using near infrared-absorbing nanoparticles”, Cancer Letters 209 (2004), pp. 171–176.
8. http://www.cancer.gov/cancertopics/factsheet/Therapy/hyperthermia
9. http://en.wikipedia.org/wiki/Ohm%27s_law
10. http://en.wikipedia.org/wiki/Joule%27s_Law
11. http://en.wikipedia.org/wiki/Minimally_invasive

Exercises

1. Build, mesh, and solve the 2D axisymmetric tumor laser irradiation model problem presented in this chapter.
2. Build, mesh, and solve the 2D axisymmetric microwave cancer therapy model problem presented in this chapter.
3. Explore other receptor materials as applied in the 2D axisymmetric tumor laser irradiation model.
4. Explore other materials as applied in the 2D axisymmetric microwave cancer therapy model.
5. Explore the different geometries in the 2D axisymmetric tumor laser irradiation model.
6. Explore the different geometries in the 2D axisymmetric microwave cancer therapy model.
7. Explore the different tissues in the 2D axisymmetric tumor laser irradiation model.
8. Explore the different tissues in the 2D axisymmetric microwave cancer therapy model.

Index

Italicized page locators indicate a figure; tables are noted with a *t*.

C

E

F

I

J

K

L

M

P

Q

R

S